



UNIVERSITÀ DEGLI STUDI DI PADOVA
DIPARTIMENTO DI INGEGNERIA CIVILE EDILE E
AMBIENTALE ICEA

Corso di Laurea in Ingegneria Civile

Tesi di Laurea Magistrale

**A Pre-processor for Numerical Analysis of
Cross-Laminated Timber Structures**

LAUREANDO:

Alessandra Ferrandino

RELATORE:

Prof. Ing. Roberto Scotta

CO-RELATORE:

Dr. Antonia Larese De Tetto

ANNO ACCADEMICO 2014 - 2015

ABSTRACT

Cross-laminated timber, also known as X-Lam or CLT, is well established in Europe as a construction material. Recently, implementation of X-Lam products and systems has begun in countries such as Canada, United States, Australia and New Zealand. So far, no relevant design codes for X-Lam construction were published in Europe, therefore an extensive research on the field of cross-laminated timber is being performed by research groups in Europe and overseas. Experimental test results are required for development of design methods and for verification of design models accuracy.

This thesis is part of a large research project on the development of a software for the modelling of CLT structures, including analysis, calculation, design and verification of connections and panels. It was born as collaboration between Padua University and Barcelona's CIMNE (International Centre for Numerical Methods in Engineering). The research project started with the thesis "*Una procedura numerica per il progetto di edifici in Xlam*" by Massimiliano Zecchetto, which develops a software, using MATLAB interface, only for 2D linear elastic analysis. Follows the phase started in March 2015, consisting in extending the 2D software to a 3D one, with the severity caused by modelling in three dimensions. This phase is developed as a common project and described in this thesis and in "*An algorithm for numerical modelling of Cross-Laminated Timber structures*" by Gabriele D'Aronco.

The final aim of the software is to enable the modelling of an X-Lam structure in the most efficient and reliable way, taking into account its peculiarities. Modelling of CLT buildings lies into properly model the connections between panels. Through the connections modelling, the final aim is to enable the check of preliminarily designed connections or to find them iteratively, starting from hypothetical or random connections.

This common project develops the pre-process and analysis phases of the 3D software that allows the automatic modelling of connections between X-Lam panels. To achieve the goal, a new problem type for GiD interface and a new application for

KRATOS framework have been performed. The problem type enables the user to model a CLT structure, starting from the creation of the geometry and the assignation of numeric entities (beam, shell, ecc) to geometric ones, having defined the material, and assigning loads and boundary conditions. The user does not need to create manually the connections, as conversely needs for all commercial FEM software currently available; he just set the connection properties to the different sides of the panels. The creation of the connections is made automatically, keeping into account different typologies of connections and assembling of Cross-Lam panels. The problem type is special for X-Lam structures, meaning that all features are intentionally studied for this kind of structures and the software architecture is planned for future developments of the post-process phase.

It can be concluded that a strategy for numerical modelling of cross-laminated timber structures has been developed. Sound bases for the pre-process and analysis phases of the software have been laid. However, future research is required to develop the post-process and verification phases of the research project.

ACKNOWLEDGMENTS

First, I would like to express my sincere gratitude to my supervisor, Prof. Roberto Scotta, for giving me possibility to be part of this very interesting research project, guiding me through it with great dedication and enthusiasm. His insight, advice and ideas have been extremely valuable to the outcomes of this research project. I deeply appreciate the encouragement, availability, patience and help which he gives me. Roberto, thank you for guiding me, starting from the degree.

In turn, deep gratitude to my co-supervisor, Dr. Antonia Larese De Tetto, who guided me during my research period at Universitat Politècnica de Catalunya, sharing with me her ideas and rich experiences and providing me valuable suggestions for my research work. Antonia, thanks for all your help.

Thanks are also expressed to the staff of CIMNE, for their help with the programming codes. Especially, to Ing. Massimo Petracca, for his special and outstanding interest giving me sapient advice for this research project. Thanks to Javier Gárate Vidiella, for his invaluable help on the programming.

I would also like to express my sincere thanks to all those professors who have taught me so much, not only from a professional point of view, but also from the human point of view. I would like to thank so much Prof. Ing. Angel Carlos Aparicio Bengoechea, for believing in me from the first moment and for giving me so much strength and energy many times with a few simple words.

I am also grateful to Gabriele, who shared with me this research project. To you, thanks for your technical support and for the constant encouragement when strength and motivation to keep going were lacking.

Warm and special thanks to Barcelona, my Barcelona, for teaching me so much and turning me into what I really am today. My very deepest gratitude to my nearest: my parents and my brother, who supported me all the time during my studies, economically and morally. Many thanks to my friends, the true ones, those that there are always. Thanks to the person who more than once said me: " Fuerza mujer, tú puedes!".

I do not believe in forever, but I believe in what has been built yesterday and the day before yesterday, and in what we live today, planning tomorrow and the day after tomorrow. I would like to thank my dear friend Eddy, for having supported and support me every day in my incredible inner growth. Dear friend, thank you for your authenticity and for all those moments I saved in my heart. I do not mean many words, will never be enough; just thanks for your precious presence in my life.

And last but not least, I feel very proud and grateful to myself for everything I've accomplished, always with desire, will and all my effort. I'm proud of all my work and growth, for the constant struggle and strong perseverance to get every drop of this ocean. Even if I've had hard times, I would not change anything I've experienced in my life, everything has led me to become the person I am today. Thanks to myself because I never gave up.

Alessandra Ferrandino

Padua, 16th September 2015

AGRADECIMIENTO

En primer lugar, me gustaría expresar mi más sincero agradecimiento a mi supervisor, Prof. Ing. Roberto Scotta, por haberme dado la posibilidad de formar parte de este interesante proyecto de investigación, conduciéndome a través del mismo con gran dedicación y entusiasmo. Su perspicacia, ideas y consejos han sido extremadamente valiosos para los resultados del mismo. Aprecio profundamente toda ayuda, fomento, disponibilidad y paciencia que por su parte he recibido. Roberto, gracias por guiarme, a partir del grado.

A su vez, profunda gratitud a mi co-supervisor, Dr. Antonia Larese De Tetto, quien me guió durante mi periodo de investigación en la Universitat Politècnica de Catalunya, compartiendo conmigo sus ideas y ricas experiencias y dándome sugerencias muy valiosas para mi trabajo de investigación. Antonia, gracias por toda tu colaboración.

Muchas gracias al personal de CIMNE, por su ayuda con los códigos de programación. Sobre todo, al Ing. Massimo Petracca, por su especial y destacado interés proporcionándome sabios consejos para este proyecto investigativo. Gracias a Javier Gárate Vidiella, por su inestimable ayuda en la programación.

También me gustaría expresar mi más sincero agradecimiento a todos aquellos profesores que me han enseñado y aportado mucho, no sólo desde el punto de vista profesional, sino también desde el punto de vista humano. Gran agradecimiento al Prof. Ing. Ángel Carlos Aparicio Bengoechea, por creer en mí desde el primer momento y por transmitirme tanta fuerza y energía, incluso en ocasiones, hasta con unas simples palabras.

También agradezco a Gabriele, quien compartió conmigo este proyecto de investigación. A ti, gracias por tu apoyo técnico y por el aliento constante cuando carecía de la fuerza y la motivación para seguir adelante.

Agradecimiento cordial y especial a Barcelona, mí Barcelona! Por enseñarme tanto y convertirme en lo que realmente soy hoy. Un agradecimiento profundo a mis más queridos: mis padres y mi hermano, quienes han sido el apoyo constante,

económico y moral, durante todos mis estudios. Muchas gracias a mis amigos, los verdaderos, aquellos que están siempre en cada momento. Agradecer también a esa persona que más de una vez me dijo: “Fuerza mujer, tú puedes!”. Personalmente no creo en el “para siempre”; en cambio, creo en lo que se ha podido construir ayer y antes de ayer, y en todo aquello que vivimos hoy en día, planeando y organizando el día de mañana y de pasado mañana. Me gustaría agradecer a mi querido amigo Eddy, por haberme apoyado y continuar haciéndolo cada día en mi increíble crecimiento interior. Amigo! Gracias por tu autenticidad y por todos aquellos momentos que llevo guardados en mi corazón. Me faltarían palabras, nunca serían suficientes; simplemente gracias por tu preciosa presencia en mi vida.

Y por último, no menos importante, me siento muy orgullosa y agradecida conmigo misma, por todo lo que he logrado, siempre con muchas ganas, voluntad y todo mi esfuerzo. Me siento orgullosa de todo mi trabajo y crecimiento, por la constante lucha y fuerte perseverancia para conseguir cada gota de este océano. He tenido momentos más difíciles y duros, aun así, no cambiaría nada de lo que he vivido en mi vida; todo me ha llevado a ser la persona que hoy en día soy. Gracias a mí misma, porque nunca me di por vencida.

Alessandra Ferrandino

Padua, 16 de septiembre de 2015

TABLE OF CONTENTS

ABSTRACT	i
ACKNOWLEDGMENTS	iii
AGRADECIMIENTO	v
TABLE OF CONTENTS	vii
LIST OF FIGURES	xi
LIST OF TABLES	xix
1 CHAPTER 1: INTRODUCTION	1
1.1 Research background and motivation.....	1
1.2 Objectives and scope.....	3
1.3 Thesis structure.....	6
2 CHAPTER 2: GENERALITIES ABOUT TIMBER AND X-LAM TECHNOLOGY	9
2.1 Generalities about wood and timber.....	9
2.2 Load duration and moisture influences on timber strength.....	10
2.3 Timber classification and strength classes.....	13
2.4 Generalities about cross-laminated timber.....	17
2.5 X-Lam panels manufacturing.....	20
2.6 Advantages of X-Lam technology.....	23
2.7 X-Lam connection systems.....	27
2.7.1 Connections behaviour.....	38
2.7.2 Connections stiffness.....	38
2.7.3 Connections resistance.....	39
2.8 X-Lam structural applications.....	45

3	CHAPTER 3: GENERAL ABOUT GiD, KRATOS, TCL AND PYTHON	49
3.1	GiD processor	49
3.1.1	Interaction of GiD with the calculating module	49
3.1.2	GiD Pre-process	52
3.1.3	GiD Post-process	52
3.2	Kratos solver	53
3.2.1	Kratos advantages	55
3.3	GiD - Kratos interaction	55
3.4	Tcl language	56
3.5	Python language	58
4	CHAPTER 4: MODELLING STRATEGY OF X-LAM BUILGINGS AND CONVENTIONS	61
4.1	Modelling strategy	61
4.2	Modelling of X-Lam panels	62
4.3	Connections modelling	65
4.4	Units of measurement convention	68
4.5	Gravity convention	68
5	CHAPTER 5: PRE-PROCESSOR AND INTERFACE TUTORIAL	69
5.1	Introduction	69
5.2	Example introduction	70
5.3	Problem type selection	71
5.4	Geometry creation	73
5.5	Model properties definition	79
5.5.1	Connection properties creation	79
5.5.2	Connection properties assignation	92
5.5.3	Elements properties creation	98
5.5.4	Elements properties assignation	103

5.5.5	Loads and boundary conditions assignation	105
5.5.6	Results menu	106
5.6	Group properties definition.....	107
5.7	Material properties definition	108
5.8	Geometry meshing.....	111
5.9	File saving.....	113
5.10	Calculation.....	114
6 CHAPTER 6: PRE-PROCESSOR PROGRAMMING DETAIL ..115		
6.1	Surface creation and removal from the menu tree.....	115
6.2	Line drawing.....	116
6.3	New IDs definition	117
6.4	Files writing	118
6.4.1	Geometry info	119
6.4.2	Connection info.....	120
6.4.3	Surface element info	122
6.4.4	Line element info	122
6.4.5	More connections.....	123
6.4.6	More materials	124
6.4.7	Data orthotropic	126
6.4.8	Materials.py	127
6.5	Connections	130
6.5.1	Connections stiffness	130
6.5.1.1	“Custom-Stiffness” mode	130
6.5.1.2	“Custom-Parameters” mode.....	131
6.5.1.3	“Standard” mode	133
6.5.2	Connections resistance.....	135
6.5.2.1	“Custom-Stiffness” mode	135
6.5.2.2	“Custom-Parameters” mode.....	136
6.5.2.3	“Standard” mode	137
7 CHAPTER 7: MODELLING OF A COMPLEX STRUCTURE ..139		

7.1	Example introduction.....	139
7.2	Preliminary design phase.....	141
7.2.1	Static design of X-Lam walls and slabs.....	142
7.2.2	Seismic design of X-Lam walls and slabs.....	145
7.2.3	Equivalent static analysis.....	146
7.2.4	Connections seismic design.....	151
7.2.4.1	Shear connections.....	151
7.2.4.2	Tension connections.....	152
7.2.4.3	Connections stiffness.....	157
7.3	Modelling.....	157
7.3.1	Geometry.....	157
7.3.2	Material and elements properties.....	159
7.3.3	Connection properties.....	162
7.3.4	Boundary conditions and loads.....	165
7.4	Results.....	165
7.4.1	Structure discretization.....	166
7.4.2	Displacement field.....	166
7.4.3	Tension field.....	169
7.4.4	Reactions.....	173
8	CHAPTER 8: CONCLUSION.....	177
8.1	Main contributions.....	177
8.2	Recommendations for further research.....	178
	REFERENCES.....	181

LIST OF FIGURES

<i>Figure 1.1</i>	<i>Example of an offset surface</i>	5
<i>Figure 1.2</i>	<i>Exploded view of a panel edge</i>	5
<i>Figure 2.1</i>	<i>Effect of moisture content on flexural strength (Giordano 1993)</i>	12
<i>Figure 2.2</i>	<i>Effects of timber classification according to resistance (Piazza, Tomasi and Modena, 2007)</i>	14
<i>Figure 2.3</i>	<i>Cross-laminated timber panel (ETA-06/0138: 2006)</i>	18
<i>Figure 2.4</i>	<i>Examples of different cross-sections of X-Lam panels (ETA-06/0138: 2006)</i>	19
<i>Figure 2.5</i>	<i>Manufacturing process of X-Lam panels (FPInnovations, 2013)</i>	22
<i>Figure 2.6</i>	<i>Typical three-storey X-Lam building showing various connections between the X-Lam panels</i>	28
<i>Figure 2.7</i>	<i>Typical wall-to-foundation X-Lam connections: (a) Connection with an exposed metal plate; (b) Connection with a concealed connector; (c) Connection with a wooden profile (FPInnovations, 2013)</i>	29
<i>Figure 2.8</i>	<i>Typical parallel wall-to-wall X-Lam connections: (a) Connection with an internal spline; (b) Connection with a surface spline; (c) Connection with a half-lapped joint; (d) Tube connection system (FPInnovations, 2013)</i>	31
<i>Figure 2.9</i>	<i>Typical perpendicular wall-to-wall X-Lam connections: (a) Connection with self-tapping screws; (b) Connection with a wooden profile; (c) Connection with a metal bracket; (d) Connection with a concealed metal plate (FPInnovations, 2013)</i>	33
<i>Figure 2.10</i>	<i>Typical wall-to-floor X-Lam connections: (a) Connection with self-tapping screws; (b) Connection with a metal bracket; (c) Connection with concealed metal plates (FPInnovations, 2013)</i>	34
<i>Figure 2.11</i>	<i>Typical wall-to-floor X-Lam connections in balloon construction (FPInnovations, 2013)</i>	35
<i>Figure 2.12</i>	<i>Typical wall-to-roof X-Lam connections (FPInnovations, 2013)</i>	37

<i>Figure 2.13</i>	<i>Sihga Idefix innovative connection system (Sihga).....</i>	<i>37</i>
<i>Figure 2.14(a)</i>	<i>Failure modes for timber and panel connections with single shear (UNI-EN 1995-1-1: 2009).....</i>	<i>42</i>
<i>Figure 2.14(b)</i>	<i>Failure modes for timber and panel connections with double shear (UNI-EN 1995-1-1: 2009).....</i>	<i>42</i>
<i>Figure 2.14(c)</i>	<i>Failure modes for steel-to-timber connections (UNI-EN 1995-1-1: 2009).....</i>	<i>43</i>
<i>Figure 2.15</i>	<i>Residential and non-residential X-Lam projects: (a) 10-storey Forté in Melbourne; (b) 9-storey Stadthaus in London; (c) Open Academy in Norwich (KLH).....</i>	<i>47</i>
<i>Figure 3.1</i>	<i>Diagram of GiD workflow (GiD User Manual).....</i>	<i>51</i>
<i>Figure 3.2</i>	<i>Diagram depicting the files system (GiD User Manual).....</i>	<i>51</i>
<i>Figure 4.1</i>	<i>Orientation of local axes in wall X-Lam panels.....</i>	<i>63</i>
<i>Figure 4.2</i>	<i>Orientation of local axes in a floor X-Lam panel.....</i>	<i>63</i>
<i>Figure 4.3</i>	<i>Difference in behaviour between a real and a modelled panel.....</i>	<i>67</i>
<i>Figure 5.1</i>	<i>Problem type selection.....</i>	<i>71</i>
<i>Figure 5.2</i>	<i>GiD problem types folder.....</i>	<i>72</i>
<i>Figure 5.3</i>	<i>“Xlam-kratos” command menu.....</i>	<i>72</i>
<i>Figure 5.4</i>	<i>Gravity warning.....</i>	<i>73</i>
<i>Figure 5.5</i>	<i>“Create object” shortcut.....</i>	<i>73</i>
<i>Figure 5.6</i>	<i>First surface.....</i>	<i>74</i>
<i>Figure 5.7</i>	<i>“Copy” menu.....</i>	<i>74</i>
<i>Figure 5.8</i>	<i>“Copy” menu for copying surfaces.....</i>	<i>75</i>
<i>Figure 5.9</i>	<i>First and second surfaces.....</i>	<i>76</i>
<i>Figure 5.10</i>	<i>First, second and third surfaces.....</i>	<i>76</i>
<i>Figure 5.11</i>	<i>End geometry.....</i>	<i>77</i>

Figure 5.12	Geometry IDs	78
Figure 5.13	“Model properties” menu.....	79
Figure 5.14	Custom mode menu.....	80
Figure 5.15	“Stiffness” menu.....	81
Figure 5.16	“Connection parameters” menu.....	83
Figure 5.17	Standard mode menu.....	86
Figure 5.18	“Connection” menu.....	86
Figure 5.19	Hold-down database	87
Figure 5.20	Brackets and distributed nailings database.....	88
Figure 5.21	"Property1" parameters.....	89
Figure 5.22	"Property2" parameters.....	90
Figure 5.23	Custom menu with all properties	91
Figure 5.24	Custom menu with all properties and their mode.....	91
Figure 5.25	“Surfaces” menu.....	92
Figure 5.26	Geometry created before.....	93
Figure 5.27	“Delete” shortcut.....	93
Figure 5.28	End “Surfaces” menu	94
Figure 5.29	End geometry	95
Figure 5.30	Selection of a connection property	96
Figure 5.31	Drawing of line 2	96
Figure 5.32	“Surfaces” menu with assigned connection properties.....	97
Figure 5.33	Elements “Properties” menu.....	98
Figure 5.34	Beam property parameters	99
Figure 5.35	Solid property parameters	99

<i>Figure 5.36</i>	<i>Shell property parameters.....</i>	<i>100</i>
<i>Figure 5.37</i>	<i>Elements “Properties” end menu</i>	<i>102</i>
<i>Figure 5.38</i>	<i>“Elements” menu.....</i>	<i>103</i>
<i>Figure 5.39</i>	<i>Beam element assignation.....</i>	<i>103</i>
<i>Figure 5.40</i>	<i>Shell thick element assignation.....</i>	<i>104</i>
<i>Figure 5.41</i>	<i>Solid element assignation</i>	<i>104</i>
<i>Figure 5.42</i>	<i>“Elements” end menu.....</i>	<i>105</i>
<i>Figure 5.43</i>	<i>“Loads” and “Boundary conditions” menu.....</i>	<i>105</i>
<i>Figure 5.44</i>	<i>“Results” menu.....</i>	<i>107</i>
<i>Figure 5.45</i>	<i>“Groups” menu</i>	<i>108</i>
<i>Figure 5.46</i>	<i>“Materials” menu.....</i>	<i>109</i>
<i>Figure 5.47</i>	<i>Material sub-menu.....</i>	<i>110</i>
<i>Figure 5.48</i>	<i>Material creation, removal and renaming window</i>	<i>111</i>
<i>Figure 5.49</i>	<i>“Mesh generation” window.....</i>	<i>112</i>
<i>Figure 5.50</i>	<i>Structured mesh selection</i>	<i>112</i>
<i>Figure 5.51</i>	<i>Meshed geometry</i>	<i>113</i>
<i>Figure 6.1</i>	<i>“Draw line” section of the code</i>	<i>116</i>
<i>Figure 6.2</i>	<i>“Draw line” groups and geometry.....</i>	<i>117</i>
<i>Figure 6.3</i>	<i>“More-Materials” file</i>	<i>125</i>
<i>Figure 6.4</i>	<i>“Materials.py” first section: list of elements properties</i>	<i>127</i>
<i>Figure 6.5</i>	<i>“Materials.py” added section: “dataOrtho” file reading.....</i>	<i>128</i>
<i>Figure 6.6</i>	<i>“Materials.py” added section: matrix creation.....</i>	<i>129</i>
<i>Figure 6.7</i>	<i>“Materials.py” added section: composite cross section creation.....</i>	<i>129</i>
<i>Figure 6.8</i>	<i>Stiffness calculation for “Custom-Parameters” mode</i>	<i>132</i>

Figure 6.9	Values set in the code for the hold-down WHT340 $\Phi 4*40$ with total nailing	134
Figure 6.10	Values set in the code for the bracket WBR100 $\Phi 4*60$ with total nailing, concrete-timber	134
Figure 6.11	Values set in the code for the distributed nailing with nail diameter of 4 mm and nail length of 30 mm.....	134
Figure 6.12	Stiffness calculation for “Standard” mode.....	134
Figure 6.13	Design load-carrying capacities calculation for “Custom-Stiffness” mode	135
Figure 6.14	Design load-carrying capacities calculation for “Custom-Parameters” mode.....	136
Figure 6.15	Design load-carrying capacities calculation for “Standard” mode	138
Figure 7.1(a)	Building front views A and B	139
Figure 7.1(b)	Building front views C, 1, 2 and 3	140
Figure 7.2(a)	Ground floor plan	140
Figure 7.2(b)	First floor plan.....	141
Figure 7.3	Loads analysis.....	142
Figure 7.4	Loads representation on the geometry.....	143
Figure 7.5	Preliminary design table for single-span floors (Rothoblaas)	144
Figure 7.6	Preliminary design table for external walls (Rothoblaas).....	145
Figure 7.7	Afferent heights of the slabs.....	146
Figure 7.8	Elastic response spectrum for Gerona, Friuli Venezia Giulia, Italy	148
Figure 7.9	Barycentre position at the two storeys of the building	150
Figure 7.10	Selected shear connection (Rothoblaas).....	151
Figure 7.11	Selected tension connections (Rothoblaas).....	153
Figure 7.12	Forces causing the first floor rigid rotation	154

<i>Figure 7.13(a) Scheme of walls and relative forces in x-direction</i>	<i>156</i>
<i>Figure 7.13(b) Scheme of walls and relative forces in y-direction</i>	<i>156</i>
<i>Figure 7.14(a) Building geometry in GiD (a)</i>	<i>158</i>
<i>Figure 7.14(b) Building geometry in GiD (b)</i>	<i>158</i>
<i>Figure 7.14(c) Building geometry in GiD (c)</i>	<i>159</i>
<i>Figure 7.15(a) Floors shell elements</i>	<i>160</i>
<i>Figure 7.15(b) Walls shell elements</i>	<i>160</i>
<i>Figure 7.16(a) Curbs beam elements</i>	<i>161</i>
<i>Figure 7.16(b) Ordinary beam elements</i>	<i>161</i>
<i>Figure 7.17 Connection property of line 36 belonging to surface 6</i>	<i>163</i>
<i>Figure 7.18 Example of line (36) belonging to wall (surface 6) and slab (surface 31); Example of lines (4 and 5) belonging to walls (surfaces 4 and 5) which are actually a single wall</i>	<i>164</i>
<i>Figure 7.19 Connection properties of lines 4 and 5 belonging respectively to surfaces 4 and 5</i>	<i>164</i>
<i>Figure 7.20 Meshed view of the structure in GiD</i>	<i>166</i>
<i>Figure 7.21 X-displacement contour [m] for the configuration A</i>	<i>167</i>
<i>Figure 7.22 Z-displacement contour [m] for the configuration A</i>	<i>167</i>
<i>Figure 7.23 Contour of the absolute value of the displacement [m] for the configuration A, taking into account only the storeys seismic forces ...</i>	<i>168</i>
<i>Figure 7.24 Y-displacement contour [m] for the configuration B</i>	<i>168</i>
<i>Figure 7.25 Contour of the absolute value of the displacement [m] for the configuration B, taking into account only the storeys seismic forces ...</i>	<i>169</i>
<i>Figure 7.26 Shell Force Sxx contour [N/m] for the configuration A</i>	<i>170</i>
<i>Figure 7.27 Shell Force Syy contour [N/m] for the configuration A</i>	<i>170</i>
<i>Figure 7.28 Shell Force Szz contour [N/m] for the configuration A, taking into account only the storeys seismic forces</i>	<i>171</i>

Figure 7.29	Shell Force S_{yy} contour [N/m] for the configuration B	171
Figure 7.30	Shell Force S_{xx} contour [N/m] for the configuration B	172
Figure 7.31	Shell Force S_{zz} contour [N/m] for the configuration B, taking into account only the storeys seismic forces	172
Figure 7.32	Z-Reaction vectors display for the configuration A.....	173
Figure 7.33	Z-Reaction vectors display for the configuration A, taking into account only the storeys seismic forces	173
Figure 7.34	Z-Reaction vectors display for the configuration B.....	174
Figure 7.35	Z-Reaction vectors display for the configuration B, taking into account only the storeys seismic forces	174
Figure 7.36	X-Reaction vectors display for the configuration A.....	175
Figure 7.37	Y-Reaction vectors display for the configuration B.....	175

LIST OF TABLES

Table 2.1	<i>Load duration classes according to UNI EN 1995-1: 2009</i>	11
Table 2.2	<i>Values of k_{mod} according to UNI EN 1995-1: 2009</i>	13
Table 2.3	<i>Recommended partial factors γ_M for material properties and resistances according to UNI EN 1995-1: 2009</i>	13
Table 2.4	<i>Strength classes according to EN 338: 2004 for solid wood of conifers and poplar</i>	16
Table 2.5	<i>Strength classes according to EN 338: 2004 for hardwood (poplar excluded)</i>	16
Table 2.6	<i>Values of K_{ser} for fasteners and connectors in N/mm in timber-to-timber and wood-based panel-to-timber connections, according to UNI-EN 1995-1-1: 2009 (the density ρ_m is expressed in kg/m³ and the diameter is expressed in mm)</i>	39
Table 6.1	<i>“Geometry-info” file</i>	119
Table 6.2	<i>“Connection-info” file</i>	121
Table 6.3	<i>“Surface-element-info” file</i>	122
Table 6.4	<i>“Line-element-info” file</i>	123
Table 6.5	<i>“More-Connections” file</i>	124
Table 6.6	<i>“Data-Orthotropic” file</i>	126
Table 6.7	<i>“Connection-info” file for line 7</i>	131
Table 6.8	<i>“Connection-info” file for line 11</i>	133
Table 6.9	<i>“More-Connections” file for line 7</i>	136
Table 6.10	<i>“More-Connections” file for line 11</i>	137
Table 7.1	<i>Design concept, structural types and upper limit values of the behaviour factors for the three ductility classes (EN 1998-1: 2004)</i>	147
Table 7.2	<i>Storeys seismic forces</i>	150

<i>Table 7.3(a) Summary of forces, stresses and hold-down disposed on walls in x-direction</i>	<i>155</i>
<i>Table 7.3(b) Summary of forces, stresses and hold-down disposed on walls in y-direction</i>	<i>155</i>

CHAPTER 1

INTRODUCTION

1.1 Research background and motivation

Wood as a building material possesses some inherent characteristics that make timber structures particularly suited for the use in regions with a high seismic risk, both due to material properties, such as lightness and load bearing capacity (good weight-to-strength-ratio), and to system properties, like ductility and energy dissipation. Recently, there have been new developments with prefabricated timber elements, which aim to address modern building requirements for cost, constructability and structural performance. Massive cross-laminated timber panels (X-Lam), which can be used as wall panels, floor panels or roof panels in timber buildings, are becoming a stronger and economically valid alternative to traditional masonry or concrete buildings in Europe, and recently also overseas. Especially in seismic-prone countries, X-lam buildings are gaining more and more popularity. However, due to relatively short time since this wood engineered product has been launched to the market, the knowledge about cross-lam as a structural material is still limited. In recent years, several research projects around Europe and in North America have been launched, with an aim to better understand the potential of cross-lam technology as a seismic resistant construction system.

Still limited is also the knowledge about the modelling of X-Lam structures, reason why a large research project started to investigate the development of a software for the analysis, calculation, design and verification of X-Lam structures. This project was born as collaboration between Padua University and Barcelona's CIMNE (International Centre for Numerical Methods in Engineering). Modelling of CLT buildings lies into properly model the connections between panels; they play an essential role in maintaining the integrity of the timber structure and providing strength, stiffness, stability and ductility to the structure. The connections may be modelled with

punctual or distributed spring elements, or with shell elements. Anyway the goal is to provide the needed flexibility to the connecting points, to avoid a fully unreal behaviour of the building, being the panels very rigid in comparison to the anchoring connections. Through the connections modelling, the final aim is to enable the check of preliminarily designed connections or to find them iteratively, starting from hypothetical or random connections.

The research project started with the thesis “*Una procedura numerica per il progetto di edifici in Xlam*” by Massimiliano Zecchetto, which develops a software, using MATLAB interface, only for 2D linear elastic analysis. Follows the phase started in March 2015, consisting in extending the 2D software to a 3D one, with the severity caused by modelling in three dimensions. This phase is described in this thesis and in “*An algorithm for numerical modelling of Cross-Laminated Timber structures*” by Gabriele D’Aronco; it consists in the pre-process and analysis phases of the 3D software. Further research is still needed to develop the post-process and verification phases.

The development of this research project arises from the need to model and analyse an X-Lam structure in the most efficient and reliable way, taking into account its peculiarities. Proper modelling strategy results in the development of a special software. This comes from the non-adaptability to X-Lam technology of the established procedures for numerical modelling adopted for other types of buildings. Nowadays the commercial FEM software available do not provide an automatic way to model a CLT structure. All software, regardless of the strategy chosen for modelling the connections, only enable to model them manually. For instance, if they are modelled with punctual springs, the user needs to duplicate the nodes and create the spring elements one by one at the pre-process interface. Follows that, if the structure is big and complex as it can be a real one, the use of these software could require time and cost expenditure and it may cause several errors because of its complexity: hundreds or thousands, if not more, may be the nodes, elements and properties that should be assigned. The aim of this research project is exactly to provide a software that allows the automatic modelling of connections between X-Lam panels, trying to avoid the human error and the cost in doing it manually.

The most convenient strategy for modelling X-Lam structures has to be defined. Such strategy must be suitable for automatic generation of numerical models and must have the ability of keeping into account all the possible typologies of connections and assembling of Cross-Lam panels. In view of future evolution of the research, the possibility of non-linear behaviour of joints and optimal automatic design via iterative solutions has to be accomplished too.

1.2 Objectives and scope

The focus of this thesis is on the continuation of the research project on the development of a software for the modelling of CLT structures, including analysis, calculation, design and verification of connections and panels. The research work will include the pre-process phase and the analysis one, the first of which is discussed in this thesis, the second one in “*An algorithm for numerical modelling of Cross-Laminated Timber structures*” by Gabriele D’Aronco.

The procedure is developed using GiD as interface support and processor and KRATOS Multiphysics as FEM framework. Relatively to the pre-process phase, the work involved in programming and numerical implementation of the interface and the data needed for the analysis, by creating a problem type. Relatively to the analysis phase, the work consisted in the development of the whole procedure of connections modelling, by creating a new application in Kratos.

Considering the numerical and computational aspects of X-Lam structures, several are the limits and issues that make it difficult to create models fully representative of their real behaviour.

Cross-Lam wall panels are very rigid in comparison to the anchoring connections, so most of the flexibility is concentrated precisely in the latter. To correctly model the building, avoiding to make it too rigid, the connections are modelled with punctual spring elements. They enable to simulate the behaviour of the different kind of connections available in X-Lam technology.

Additionally, a limit lies in the behaviour of the spring elements to use in the modelling. The behaviour of a CLT structure in static conditions can be assimilated to a contact problem because the walls are supported all along their lower side by the soil. In this condition, the walls only work to compression, they do not offer any resistance to tension. The possible lifting of the walls, which may occur in seismic conditions, is resisted by the hold-down connections that offer the tension resistance to the walls. To simulate properly the contact problem, the springs should present a non-linear constitutive law in axial direction. Alternatively, the problem can be considered non-linear for the material, considering the hold-down as a material resistant to compression and the soil as a material non-reactive to traction. Within this project, the springs present a linear elastic constitutive law, leading to the need of modifying the hold-down stiffness compared to the real one. This is to take into account the difference in behaviour, under the action of horizontal forces, of a single modelled panel compared with the same in a real situation. Therefore, being the spring elements currently added in Kratos only implemented with linear elastic constitutive law, the analysis is always considered linear elastic.

In reference to the behaviour of a single X-Lam panel, this is an orthotropic rather than an isotropic material. This is due to the different total thickness of the layers in longitudinal and transversal direction and to the difference in value of the elastic modulus of the timber, which is one order of magnitude greater in the direction parallel to the grain than in the transversal direction. These two topics lead to adopt an elastic orthotropic constitutive law for the shell elements.

The research work developed in this common project concerns the creation of a new problem type, especial for X-Lam structures. It enables the user to model a CLT structure, starting from the creation of the geometry and the assignation of numeric entities (beam, shell, ecc) to geometric ones, having defined the material, and assigning loads and boundary conditions. The user does not need to create manually the connections, he just set the connection properties to the different sides of the panels. Also the punctual connections (hold-down) are assigned at the interface to the lines; conversely, in the analysis they are assigned only to the extreme points of the panel side.

The creation of the connections is made automatically: an abstract offset is applied between each surface and line, or, better, between each border shell in which the surface is discretized and beam elements. The information about the connection property is stored at interface level to the line (geometric entity), which is discretized, depending on the mesh, in one or more beams (numeric entities). The offset (Figure 1.1) implies the duplication of the nodes that belong to both beams and shells. It has zero distance to allow an easy management of the nodes, since the duplicated nodes will have the same coordinates of the original ones.

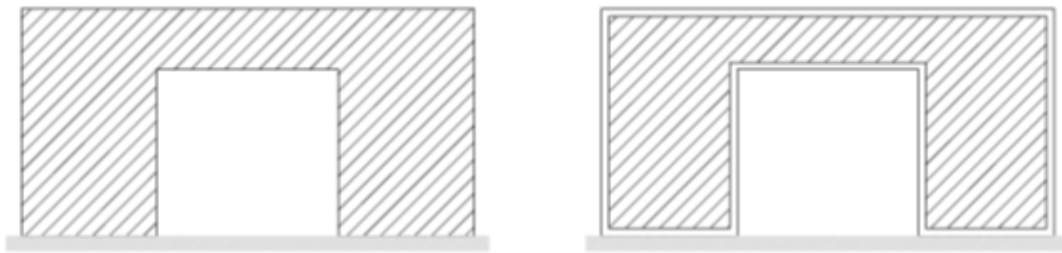


Figure 1.1 Example of an offset surface

Therefore, spring elements, with the stiffness values inserted by the user at the interface, are used to join nodes with equal coordinates. The beam elements, necessary for the duplication of the nodes, are considered fake elements, if not set as curbs at the interface. Their geometric and structural properties are so that their presence is negligible in the analysis of the structure. Figure 1.2 shows the springs connecting the shells and beams.

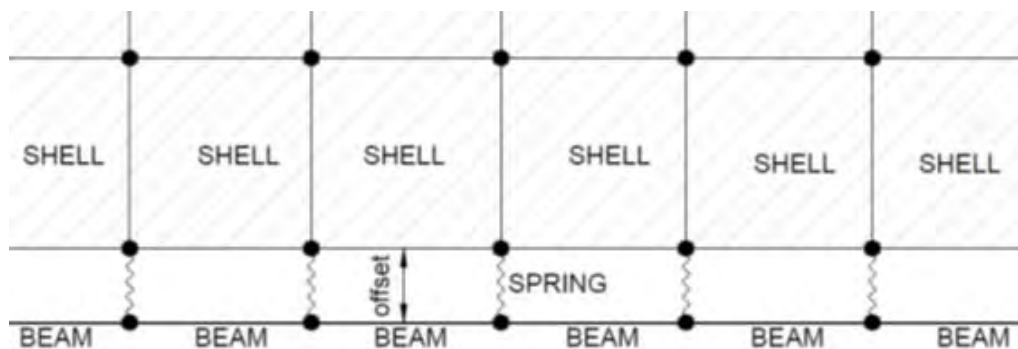


Figure 1.2 Exploded view of a panel edge

The pre-process phase, concerning the creation of a new problem type in GiD, is described in this thesis. It enables, at interface level, the creation of geometry and the

assignment of elements properties, material, loads, boundary conditions and, above all, connection properties. Moreover, it allows the creation of suitable input data files for the analysis and the modelling of the panels as orthotropic shells with composite cross section.

The analysis phase, concerning the numerical changes in Kratos framework, is described in the thesis “*An algorithm for numerical modelling of Cross-Laminated Timber structures*” by Gabriele D’Aronco. It consists in the implementation of spring elements and the numerical procedure for automatic modelling of the connections, meaning duplication of panels’ border nodes and joint of nodes by means of spring elements.

1.3 Thesis structure

A brief summary of each chapter of the thesis is given in this Section. In each chapter, the first section overviews general information about the chapter topic; in subsequent sections, theoretical and numerical investigations are described.

Chapter 2 provides an overview of general information about wood, timber and cross-laminated timber technology. First, influences on timber strength and timber classification are introduced. Then, description of cross-lam panels and typical X-Lam connection systems are presented. A state-of-the-art of cross-lam timber application is highlighted at the end of this Chapter.

Chapter 3 provides an overview of general information about the processor, the solver and the programming languages used for the thesis: GiD, Kratos, Tcl and Python. First, description of GiD pre-process and post-process and Kratos tools and advantages is introduced, up to their interaction; then, a brief description of Tcl and Python features is presented.

Chapter 4 provides a description of the strategy adopted for modelling Cross-Lam buildings, explaining in detail panels and connections modelling. At the end of the Chapter, significant conventions assumed in the modelling are particularised.

Chapter 5 provides an extensive explanation of the pre-processor at interface level. It is a user tutorial, starting from the problem type selection and geometry creation up to the model properties and material definition, and the calculation of the desired structure.

Chapter 6 provides an extensive explanation of the pre-processor at programming level. It presents the code implemented to obtain all the tools of the pre-process, with a detailed description of the files generated by the processor, the composition of the shell cross section and the calculation of the connections stiffness and resistance.

Chapter 7 provides a presentation of the modelling of an X-Lam structure case study, starting from the preliminary design phase up to the actual modelling in GiD, especially from an engineering point of view. Finally, the results of the analysis are displayed.

CHAPTER 2

GENERALITIES ABOUT TIMBER AND X-LAM TECHNOLOGY

This Chapter presents the main characteristics of wood and timber, providing a classification according to resistance. Later, general information about cross-laminated timber technology is displayed. Typical X-Lam connection systems are detailed, focusing specifically on their stiffness and capacity calculation. Finally, state-of-the-art of cross-lam timber application is highlighted.

2.1 Generalities about wood and timber

The wood mechanical properties are intimately related to the natural origin of the material. Cell morphology guarantees high resistance values with low self-weight. The cellular organization of the wood is however also at the origin of a marked anisotropy of the mechanical properties of the material, and this results in a marked difference of the stiffness and resistance values, according to the direction of the applied load or, in a dual way, depending on the grain direction. Wood is more resistant and rigid to stresses oriented along the grain direction.

Conversely, solid timber in structural dimensions is a non-homogeneous material, which contains defects related to the growth of the plant from which it comes, in the form of nodes, localized grain deviations and many others. These defects significantly reduce the resistance when the wood is sawn and used for other uses. Therefore it is evident that the mechanical characteristics of the structural timber cannot be derived from those of the net wood without taking into account the defects. Besides the presence of defects within the wood mass, the study of the timber subjected to external stresses is complicated by the strong influence of moisture variation and load duration on the resistance.

2.2 Load duration and moisture influences on timber strength

Firstly, strength is affected by load duration. For timber, as for all construction materials, the resistance to short term loads is higher than for long term loads. Additionally, studies conducted by Madsen have shown that load duration influence depends on the timber quality and it is significantly lower for low qualities than high ones. In principle, this can be explained by the fact that for lower qualities the knots determine the resistance, while for higher qualities it is the wooden base quality to be decisive. Thus, for higher classes the dependence of the resistance on the load duration will be similar to that of the net wood, while for lower classes the high defectiveness will be so decisive as to reduce the contribution to the decrease of resistance for the load duration due to the base material.

Indeed, the presence of knots generates, for short duration loads, strong peaks of tension concentration (elasticity of the material) that will determine the level of resistance in the short term. Conversely, for long term loads the resistance relative to the timber itself would tend to decrease, but the concentrations of tensions around knots (viscosity of the material) tend to be blunted by acting in a manner favourable to the resistance. Therefore, thanks to these two factors counteracting each other, the gap between resistance for short and long term loads gets down.

Moreover, the dependence on the moisture content should be considered. Indeed, it was found that, for the worst qualities (meaning low resistance values), the influence of the moisture content in the timber appears more limited than in net wood. The influence of moisture content on the reduction of resistance results less important the poorer the material. Potentially, the high defectiveness becomes so crucial to flatten the differences in resistance between the dry base material and the same wet base material.

The European regulations (UNI EN 1995-1: 2009) take into account these influences defining for loads the so-called "load duration classes", and for hygrometric conditions the so-called "humidity classes" or " service classes".

The loads are distinguished in permanent loads (always of long duration, of course) and accidental loads, which may be of long, medium, short term or instantaneous; they are shown in Table 2.1.

Table 2.1 Load duration classes according to UNI EN 1995-1: 2009

Load-duration class	Order of accumulated duration of characteristic load
Permanent	more than 10 years
Long-term	6 months – 10 years
Medium-term	1 week – 6 months
Short-term	less than one week
Instantaneous	

Generally, permanent loads are represented by the weight of the structure, medium term loads are loads imposed to the slabs (for example, overload for residential use), whereas snow and wind contribute to short-term, earthquakes to instantaneous loads.

To take into account changes in moisture in the timber, the code prescribes that the constructions are assigned, depending on the thermo-hygrometric environment, to one of the following service classes:

- Service class 1: this class is characterised by a content of moisture in the materials corresponding to a temperature of $20 \pm 2^{\circ}\text{C}$ and at a relative humidity of the surrounding air which exceeds 65% only for a few weeks per year. In the service class 1 the timber average humidity, in the majority of conifers, is not greater than 12%.
- Service class 2: this class is characterised by a content of moisture in the materials corresponding to a temperature of $20 \pm 2^{\circ}\text{C}$ and at a relative humidity of the surrounding air which exceeds 80% only for a few weeks per year. In the service class 2 the timber average humidity, in the majority of conifers, is not greater than 20%.
- Service class 3: this class includes all climatic conditions which give rise to higher moisture content in the timber.

To service class 2 belong buildings heated in a non-continuous way and ventilated, such as houses for leisure, garages without heating, warehouses and cellars.

To service class 3 belong buildings exposed to rainfall or in any case to water, including the shuttering for the concrete and scaffolding outdoors, as well as support structures of the roofs not adequately insulated.

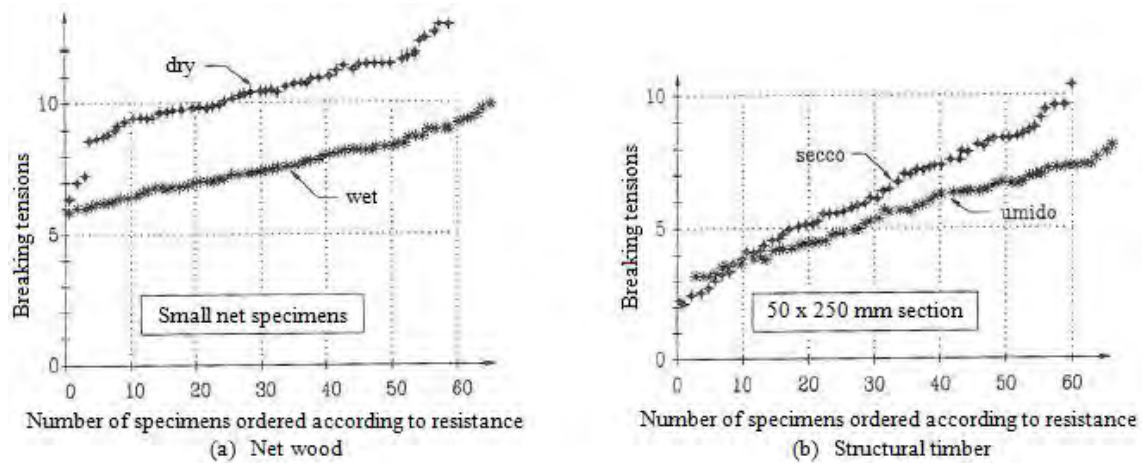


Figure 2.1 *Effect of moisture content on flexural strength (Giordano, 1993)*

The code UNI EN 1995-1-1: 2009 defines then a correction factor K_{mod} that takes into account the effect, on the strength parameters, of both load duration and moisture content of the structure. In case a combination load includes actions belonging to different classes of load duration, the code requires the choice of a K_{mod} value that corresponds to the shorter duration action. The values for the correction factor K_{mod} recommended by the code are given in Table 2.2; values are limited to solid timber, glued laminated timber (to which they refer X-Lam panels) and certain other products based on timber.

The design value of a generic property of the material can be defined as:

$$X_d = K_{mod} \frac{X_k}{\gamma_M}$$

being γ_M the partial safety factor for a given material property, X_k the characteristic value of the same property and X_d its design value. The values for the partial factors γ_M recommended by the code UNI EN 1995-1-1: 2009 are given in Table 2.3.

Table 2.2 Values of k_{mod} according to UNI EN 1995-1: 2009

Material	Service class	Load-duration class				
		Permanent	Long term	Medium term	Short term	Instantaneous
Solid timber	1	0,60	0,70	0,80	0,90	1,10
	2	0,60	0,70	0,80	0,90	1,10
	3	0,50	0,55	0,65	0,70	0,90
Glued Laminated timber	1	0,60	0,70	0,80	0,90	1,10
	2	0,60	0,70	0,80	0,90	1,10
	3	0,50	0,55	0,65	0,70	0,90
LVL	1	0,60	0,70	0,80	0,90	1,10
	2	0,60	0,70	0,80	0,90	1,10
	3	0,50	0,55	0,65	0,70	0,90
Plywood	1	0,60	0,70	0,80	0,90	1,10
	2	0,60	0,70	0,80	0,90	1,10
	3	0,50	0,55	0,65	0,70	0,90
Fibreboard, MDF	1	0,20	0,40	0,60	0,80	1,10
	2	—	—	—	0,45	0,80

Table 2.3 Recommended partial factors γ_M for material properties and resistances according to UNI EN 1995-1: 2009

Fundamental combinations:	
Solid timber	1,3
Glued laminated timber	1,25
LVL, plywood, OSB,	1,2
Particleboards	1,3
Fibreboards, hard	1,3
Fibreboards, medium	1,3
Fibreboards, MDF	1,3
Fibreboards, soft	1,3
Connections	1,3
Punched metal plate fasteners	1,25
Accidental combinations	1,0

2.3 Timber classification and strength classes

To allow a secure and reliable design of the timber structural elements, the characteristics of the material must be known with sufficient reliability. The mechanical characteristics of the wood show a very large dispersion of values; for example the ratio between the smallest and the largest value of the failure resistance of a sawn wood element can reach 1:10. This would prevent, in absence of an effective classification,

the use of timber as a structural element properly. The procedure for the classification of the material is intended to achieve the following goals (with reference to Figure 2.2):

- determination of classes of resistance with differentiated properties and reliable characteristic values;
- smaller dispersion of the values of the mechanical properties inside of each class of resistance compared to the totality of the material (this effect is defined "homogenisation" of the material).

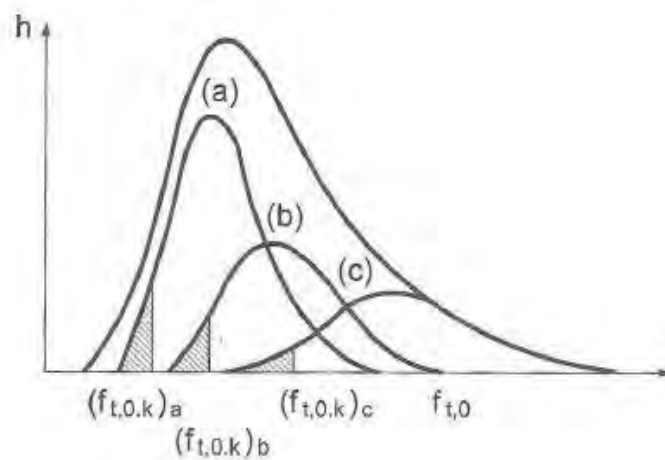


Figure 2.2 *Effects of timber classification according to resistance (Piazza, Tomasi and Modena, 2007)*

Numerous and complexes are the factors that can affect the resistance of timber elements, leading the regulators to adopt an approach consisting in the following points:

- selection of elements suitable for structural use and having minimum physical-mechanical guaranteed characteristics (classification according to resistance);
- assignment to classified elements of characteristic values of the main mechanical properties (strength classes and characteristic performance profiles);
- design of the elements by means of calculation rules specifically designed to use these characteristic values.

The method used to define the properties of the timber elements for structural purposes in the European legislation is the semi-probabilistic limit state. In order to fall under this methodology it is necessary to abandon the mechanical characterisation of the

net wood and enlist to a mechanical description of the structural element for a given applied stress.

The characteristic values of strength and modulus of elasticity are therefore defined as values to the 5th percentile of the population obtained from tests with loading time of about 300 seconds on specimens under normal conditions.

The performance of a structural element in timber, as already briefly noted, are strongly affected by the presence and position, in the same element, of some natural features that, in structural field, are called defects. For this reason, the structural design needs a classification according to resistance of the structural element in the dimensions of use, rather than on the net material, for both elements in laminated wood and hardwood.

Tables 2.4 and 2.5, adapted from the code EN 338: 2004, show the strength classes and the related characteristic values of the main properties, for coniferous wood, poplar wood and hardwood in general. With reference to the glue laminated timber and X-Lam panels, these tables give the values of some mechanical properties relative to the slats they contain.

It is clear that the performance characteristics of the finished product, for example the bending resistance in glulam beams, are related to the resistance characteristics of the individual boards, as well as, of course, to other aspects related to the proper execution of butt joints and the bonding between overlapping slats. As concern the glue-laminated timber elements, excluding the X-Lam panels, the approach of the new European legislation is to suggest the manufacturer the mechanical features of the starting sawn to use, to get a glued laminated timber element belonging to a particular class of resistance. The standard EN 1194: 1999, in particular, provides a set of relations, here omitted for brevity, for calculation of mechanical properties of timber laminated elements according to resistance properties of the individual slats.

Table 2.4 Strength classes according to EN 338: 2004 for solid wood of conifers and poplar

Characteristic values	Classes											
	C14	C16	C18	C20	C22	C24	C27	C30	C35	C40	C45	C50
Resistances [MPa]												
Bending	14	16	18	20	22	24	27	30	35	40	45	50
Tension parallel to grain	8	10	11	12	13	14	16	18	21	24	27	30
Tension orthogonal to grain	0,4	0,5	0,5	0,5	0,5	0,5	0,6	0,6	0,6	0,6	0,6	0,6
Compression parallel to grain	16	17	18	19	20	21	22	23	25	26	27	29
Compression orthogonal to grain	2,0	2,2	2,2	2,3	2,4	2,5	2,6	2,7	2,8	2,9	3,1	3,2
Shear	1,7	1,8	2,0	2,2	2,4	2,5	2,8	3,0	3,4	3,8	3,8	3,8
Elastic modulus [GPa]												
Mean, parallel to grain	7	8	9	9,5	10	11	11,5	12	13	14	15	16
Characteristic, parallel to grain	4,7	5,4	6,0	6,4	6,7	7,4	7,7	8,0	8,7	9,4	10,0	10,7
Mean, orthogonal to grain	0,23	0,27	0,30	0,32	0,33	0,37	0,38	0,40	0,43	0,47	0,50	0,53
Mean, of shear	0,44	0,50	0,56	0,59	0,63	0,69	0,72	0,75	0,81	0,88	0,94	1,00
Density [Kg/m3]												
Characteristic	290	310	320	330	340	350	370	380	400	420	440	460
Mean	350	370	380	390	410	420	450	460	480	500	520	550

Table 2.5 Strength classes according to EN 338: 2004 for hardwood (poplar excluded)

Characteristic values	Classes					
	D30	D35	D40	D50	D60	D70
Resistances [MPa]						
Bending	30	35	40	50	60	70
Tension parallel to grain	18	21	24	30	36	42
Tension orthogonal to grain	0,6	0,6	0,6	0,6	0,6	0,6
Compression parallel to grain	23	25	26	29	32	34
Compression orthogonal to grain	8,0	8,4	8,8	9,7	10,5	13,5
Shear	3,0	3,4	3,8	4,6	5,3	6,0
Elastic modulus [GPa]						
Mean, parallel to grain	10	10	11	14	17	20
Characteristic, parallel to grain	8,0	8,7	9,4	11,8	14,3	16,8
Mean, orthogonal to grain	0,64	0,69	0,75	0,93	1,13	1,33
Mean, of shear	0,60	0,65	0,70	0,88	1,06	1,25
Density [Kg/m3]						
Characteristic	530	560	590	650	700	900
Mean	640	670	700	780	840	1080

Despite X-Lam panels are produced and marketed since 1995, they have so far never been integrated in any product standards. Their use as a material for load bearing structures is therefore, to date, regulated through national approvals or by *European Technical Approval (ETA)*. The approvals contain and describe the requirements which must be imposed to the product and its production from the materials used, as well as the indications for use, dimensioning and necessary verifications. In case of European approvals there are also indications concerning the marking CE. The standard EN 16351: 2013 has been submitted to CEN members for validation and it will lead to the emergence of an EN code. A subcommittee of experts within the commission CEN TC250 is working, currently, on the integration of the X-Lam material in the UNI-EN

1995-1-1: 2009. In particular, the main features that must be evaluated for the release of ETA are:

- load-bearing capacity and stiffness relative to mechanical actions:
 - bending;
 - tension and compression;
 - shear;
 - determination of resistance to bearing stress;
- protection against noise;
- energy saving and heat retention;
- hygiene, health and environment.

Therefore this certificate, which also shows the class of the slats of which the panel is made, provides all the mechanical features necessary for the structural calculation.

2.4 Generalities about cross-laminated timber

Cross laminated timber (X-Lam or CLT) is an engineered wood product fabricated by adhering and compressing wood layers called lamellae in perpendicular grain orientations to form a solid panel. Wood layers are glued together on their wide faces and, usually, on the narrow faces as well. X-Lam technology was invented and developed in central Europe in the early 1990's and since then it has been gaining increased popularity in residential and non-residential applications. The number of buildings constructed using X-Lam panels as the main structural system has seen exponential growth in the last decade, and market share for X-Lam construction is expected to continue to escalate in the future. The European experience showed that X-Lam construction can be competitive, particularly in mid-rise and high-rise buildings due to its easy handling during construction and a high level of prefabrication. Recently, X-Lam was introduced also overseas, in North America, Australia and in New Zealand. A number of production plants have been established or they are proposed to be built in aforementioned countries.

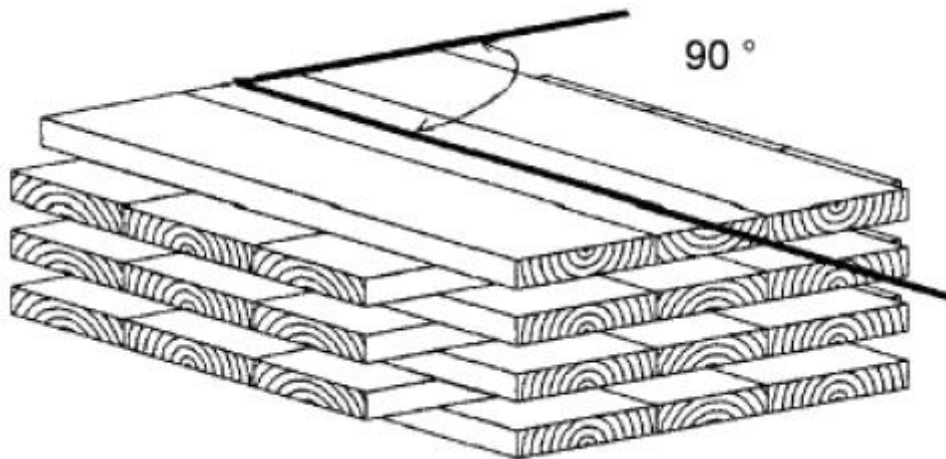


Figure 2.3 *Cross-laminated timber panel (ETA-06/0138: 2006)*

Cross-laminated timber panels are manufactured to customized dimensions; panel sizes vary by manufacturer. Lamellae thicknesses are ranging between 10 and 40 mm, and are produced of technically dried, quality-sorted and finger-jointed planks. Panel thickness is usually in the range of 50 mm to 300 mm but panels as thick as 500 mm can be produced. Production sizes range from 1.2 m to 3 m in width and 5 m to 16.5 m in length (limited by transportation restrictions or the length of a production line). The mechanical properties of X-Lam panels are provided by each producer due to the different cross section configurations and due to different properties of the single layers and boards. Openings within panels can be pre-cut in the factory to any dimension and shape, including openings for doors, windows, stairs, service channels and ducts. In order to rule out any damage caused by pests, fungi or insects, technically dried wood with an average wood moisture of 12% (+/-2%) is used to produce X-Lam solid wood panels. In plane deformation rate of X-Lam panels is about 0.01% per percentage of change in wood moisture content, while perpendicular to panel plane the deformation rate is about 0.20% per percentage of change in wood moisture content.

Typically the panels are consisted of three, five, seven or more layers of industrial dried boards, symmetrical around the mid layer. By using double layers, the longitudinal or transverse rigidity of the panel can be further enhanced. Softwood such as spruce, pine and fir is currently used in X-Lam production. Boards with different grading classes might be used for longitudinal (parallel) and transversal (perpendicular) layers to optimise mechanical and fire performances of X-Lam product. The density of a

CLT timber panel is generally around 400 to 500 kg/m³ i.e. around the density of the base laminate species used.

The external loads are carried by the longitudinal (parallel) layers, whereas the transversal (perpendicular) layers have lower strength and stiffness in the main panel direction since the stresses are perpendicular to the grains. Provided that the longitudinal layers are connected via flexible transverse layers, bending caused by transverse forces can no longer be disregarded. The so-called “rolling-shear” (shear in the radial-tangential-plane) in the transversal layers leads to relatively low load-bearing capacities. Cross-lamination in X-Lam panels have reinforcing effect for prevention from brittle failure modes such as splitting, and increases strength capacity of connections. The cross-laminating process provides improved dimensional stability to the product which allows for prefabrication of long and wide panels. Additionally, cross-laminating provides relatively high in-plane and out-of-plane strength and stiffness properties, giving it two-way action capabilities similar to a reinforced concrete slab.

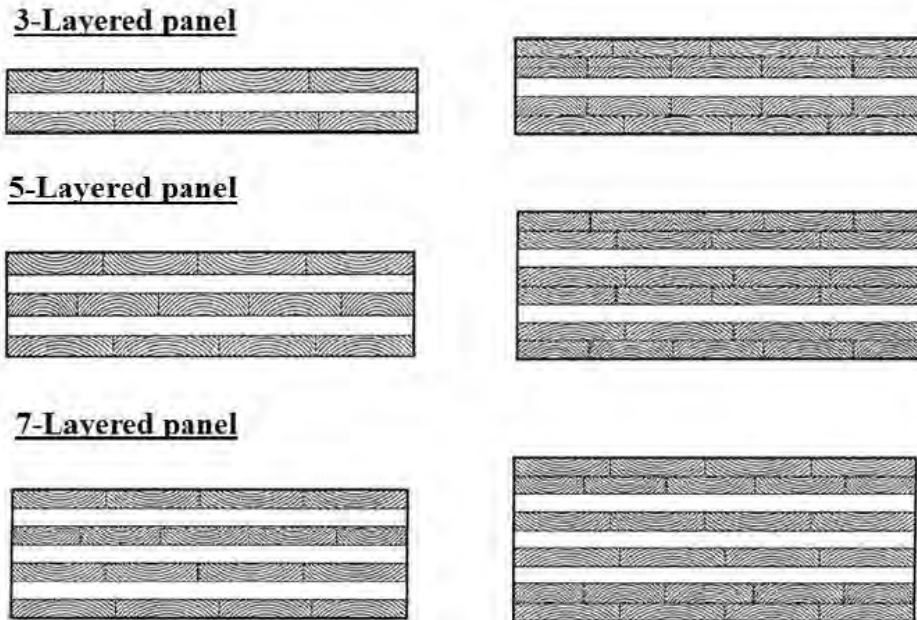


Figure 2.4 Examples of different cross sections of X-Lam panels (ETA-06/0138: 2006)

By varying the number of layers as well as the lumber species, grade and thickness, X-Lam panels can be used in various assembly types such as walls, floors,

roofs, elevator shafts, stairways etc. The wall and floor panels may be left exposed in the interior, which provides additional aesthetic attributes. The panels are used as prefabricated building components which can speed up construction practices or allow for off-site construction. While X-Lam panels act as two-way slabs, the stronger direction follows the grain of the outer layers. For example, when used for walls, X-Lam is installed so the boards on the outer layer of the panel have their grain running vertically. When panels are used in floor and roof applications, they are installed so the boards on the outer layer run parallel to the span direction.

Panels may be connected to each other with half-lapped, single or double splines made from engineered wood products. Dowel-type mechanical fasteners such as nails, screws, dowels and bolts, or bearing-type (e.g., split rings, shear plates) connectors are used to connect X-Lam panels. Typical X-Lam connections will be presented more in detail in Section 2.7 of this Chapter.

2.5 X-Lam panels manufacturing

Currently there are no standards in Europe that cover X-Lam manufacturing or installation. However, various X-Lam products have a European Technical Approval (ETA) that allows manufacturers to place CE marking. The approval process includes preparation of a European Technical Approval Guideline (ETAG) that contains specific requirements of the product as well as test procedures for evaluating the product prior to submission to the European Organization for Technical Approvals (EOTA). Finally, the ETA allows manufacturers to place CE marking (Conformité Européenne) on their products.

In the USA, the American National Standards Institute (ANSI) recently approved ANSI/APA PRG 320-2012 Standard for Performance-Rated Cross-Laminated Timber (ANSI/APA PRG 320-2012, 2012). This Standard covers manufacturing, qualification and quality assurance requirements for X-Lam products. Key stakeholders included X-Lam manufacturers, distributors, designers, users, building code regulators, and government agencies.

In general, the production of cross-laminated timber panels follows the following procedure:

- **Selection of species**

The base species of timber used for X-Lam panels depend on the region where it is manufactured. For X-Lam manufactured in Austria and Germany spruce is the main species used; pine and larch can also be used on request. X-Lam plants in Canada are likely to use S-P-F (spruce pine fir) species. Whilst production is yet to occur in Australia and New Zealand, the timber species likely to be used is radiata pine.

- **Timber laminates grouping**

Individual seasoned dimensional timbers are used, generally softwood and usually finger jointed along their length to obtain the desired lengths and quality. Individual timbers can be edged bonded together to form a timber plate before further assembly into the final panel.

- **Adhesive application**

Generally the choice of adhesives is dependent on manufacturers but the new polyurethane (PUR) adhesives are normally used as they are formaldehyde and solvent free. Occasionally, and manufacturer dependent, melamine urea formaldehyde and phenol-resorcinol-formaldehyde adhesives could be used. Both face and edge gluing can be applied.

- **Panel assembly and arrangement**

The main difference that occurs between X-Lam manufacturers is the treatment of individual layers. Some manufacturers edge bond the individual dimensional timber together to form a layer before pressing each layer into the final X-Lam panel. Other manufacturers just face bond individual dimensional timber in layers and press all of them together into the final X-Lam panel in the one operation. Panel sizes vary by manufacturer and application, as mentioned in the beginning in this Chapter.

- **Pressing**

Gluing at high pressure reduces the timbers expansion and shrinkage potential to a negligible level, thus the right pressure is essential. Hydraulic presses are normally employed, however, use of vacuum and compressed air presses is also possible, depending on panel thickness and the adhesive used. Vertical and horizontal pressings can be applied.

- **Planing and sanding**

The assembled X-Lam panels are planed or sanded for a smooth surface finish.

- **Panel final shaping**

Computer numerical controlled (CNC) routers are generally used to cut the X-Lam panel to the final length and width. Sometimes manufacturers also pre-cut openings for windows, doors and service channels, connections and ducts. The addition of insulation and exterior cladding may also take place in the factory, and the completed panels are shipped to the job site ready to be erected into place.

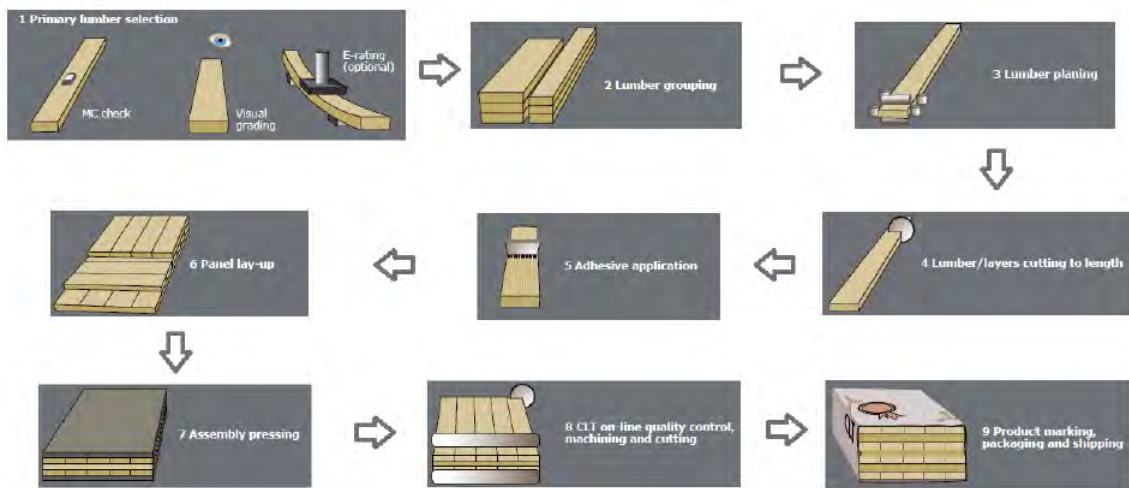


Figure 2.5 Manufacturing process of X-Lam panels (FPInnovations, 2013)

2.6 Advantages of X-Lam technology

X-Lam technology has several advantages in structural applications:

- *Low weight*

The X-Lam buildings can weigh up to four times less than its concrete counterpart, which can reduce transportation costs, allows the designers to reduce the foundation size, and eliminate the need for a tower crane during construction (Yates et al., 2008). Mobile cranes can be employed, saving substantial erection, hire and labour costs.

- *Prefabrication*

The prefabricated nature of X-Lam technology permits high precision in terms of dimensional accuracy due to CNC controlled cutting and quality controlled production. Wall, floor and roof elements can be pre-cut, including openings for doors, windows, stairs, service channels and ducts. Insulation and finishes can also be applied prior to installation, reducing demand for skilled workers on site. Construction process is characterized by increased safety on the construction site, faster project completion and availability for occupancy in a shorter time. For example, it took four carpenters just nine weeks to erect nine storeys and the entire construction process was reduced from 72 weeks to 49 weeks (Yates et al., 2008) compared to a traditional reinforced concrete building. In addition, there is less disruption to the surrounding community and less waste is produced. As most of the work occurs off-site at the factory, there is a lower demand for skilled workers on-site.

- *Easy handling and erection*

Handling the X-Lam panels requires smaller cranes which also influences on the lower cost of a building construction. One of the biggest benefits of using X-Lam panels is that the structure can be built quickly and efficiently. Because panels are designed for specific end-use applications, they are often delivered and erected using a “just-in-time” construction method, making X-Lam ideal for projects with limited on-site storage capacity. Panels are usually loaded into the truck at the manufacturing plant in the sequence that they will be required for installation on site. Where it is not possible to

install X-Lam panels immediately, they can be off-loaded and stored off the ground under a waterproof covering until required. Due to the light weight of the panels it is also common to use the building itself as a place to temporarily store panels. It is also possible to assemble elements or modules of the building off-site and deliver completed segments of the building to the site. This speeds up the construction process even further. Panels are lifted into place using pre-inserted hooks.

- *Flexibility in architectural implementation*

Versatility of X-Lam technology comes from the fact that panels can be used for many different assemblies (wall, floor, roof, stairs etc.) just by varying the thickness. X-Lam construction system can be combined also with other timber structural systems such as light timber frames, post-and-beam heavy timber system and glue-laminated timber. In addition, X-Lam elements are compatible with other building materials such as steel, concrete and glass. Compared to traditional light wood-frame construction methods, which rely on plywood sheathing over wood studs (walls) or rafters and beams (roof), the use of X-Lam panels offers an alternative in the form of a single component that is load bearing and provides an aesthetically pleasing finished surface. Depending on its intended use, X-Lam panels can be used for either visible or hidden construction applications. Its ability to be used as either a panelised or a modular system makes it ideally suited for additions to existing buildings or their upgrade. Good span-to-depth ratio allows shallow floors and slender construction elements can increase the net building area.

- *Static properties*

High in-plane and out-of-plane strength and stiffness properties of X-Lam panels enable in-plane stability of the panels and lack of susceptibility to soft-storey failures. The cross-lamination provides relatively high strength and stiffness properties in both directions, giving it a two-way action capability similar to a reinforced concrete slab.

- *Seismic performance*

In terms of seismic performance, timber buildings in general perform well because wood is relatively light as a construction material, thus inertial forces caused by

earthquakes are lower than in case of buildings made of other materials. High ductility and energy dissipation capacities of X-Lam buildings, together with sufficient strength capacity, make this construction system very effective at resisting lateral forces caused by earthquake ground motions.

- *Fire resistance*

X-Lam assemblies have inherently excellent fire-resistance due to the thickness of panels, which when exposed to fire, slow down the heat propagation within the cross section and char at a slow and predictable rate (0.67 mm/min according to ETA - 06/0138, 2006). Once formed, this char protects the wood from further degradation, helping to maintain structural integrity of the building. In addition, X-Lam structures also tend not to have as many concealed spaces within floor and wall assemblies, which reduces the risk of a fire spreading. Fire performance of X-Lam panels can also be enhanced by lining with fire resisting gypsum boards and, in case of floor panels, additional layers and coverings. A demonstration test conducted by IVALSA on a full scale, three-storey X-Lam building confirmed that X-Lam panels protected by one layer of gypsum board were able to withstand the burn out of the room contents without fire spread to adjacent rooms or floors (Frangi et al., 2008).

- *Thermal performance*

Cross-lam panels have the same fundamental thermal insulation and thermal mass properties as the wood from which they are made (thermal conductivity $\lambda = 0.13$ W/(m*K) according to EN 12524, 2000). Wood has a low thermal conductivity so reduces problems such as thermal bridging from the internal to the external environments and the other way around, thus reducing heat transfer and energy wastage.

- *Acoustic performance*

Solid wood panels offer acoustical advantages when used for floor and wall systems. When used in conjunction with insulation and gypsum board, it is possible for an X-Lam building to exceed code requirements related to the acoustical performance of floors and walls.

- *Dimensional stability*

The crosswise arrangement of the longitudinal and transverse layers reduces the swelling and shrinkage of the wood in the plane of the panel to an insignificant minimum and considerably increases the static load-carrying capacity and dimensional stability.

- *Durability*

Generally due to the quick erection time of X-Lam based systems, the short term exposure of X-Lam panels to weather is not an issue. Short term and occasional exposure to water will not have long term effect on X-Lam panels. During construction wall elements may be protected with vapour barriers or the building's scaffolding can be wrapped to form this protection. Other strategies could be employed such as coating system for the construction period only. Long-term exposure of X-Lam panels to weather is not recommended.

- *Sustainable and environmental friendly building material*

As with all wood products, the benefits of X-Lam include the fact that wood grows naturally, using solar energy and it is the only major building material that is renewable and sustainable. It also has a low carbon footprint, because the panels continue to store carbon absorbed during the tree's growing cycle and because of the greenhouse gas emissions avoided by not using products that require large amounts of fossil fuels to manufacture. Harvesting from sustainably managed forests contribute to efficient use of the resource. Many of the recent structures built from CLT benefit from these environmental considerations. For example, two mid-rise residential projects in London used the fact that wood stores carbon and that substantial greenhouse gas emission were avoided by substituting cross-lam in place of concrete or steel to get preferential approval from local planning authorities (Yates et al., 2008).

- *Recycling and reuse*

X-Lam panels can also be recycled and reused for the same or for a different purpose. Structural flexibility of the panels is very wide as well as their durability, thus enabling

panels to be reused. For example, after a series of shake table tests on a 7-storey SOFIE building in Japan, the building was disassembled and the panels were shipped back to Italy. The panels were stored for a couple of years, before they were used as main load-carrying elements in the prototype of a sustainable modular house unit made of X-Lam panels (Briani et al., 2012).

2.7 X-Lam connection systems

X-Lam wall panels are very rigid in comparison to the anchoring connections, so most of the flexibility is concentrated in the connections. Thus, connections play an essential role in maintaining the integrity of the timber structure and providing strength, stiffness, stability and ductility to the structure. The structural efficiency of the floor system acting as a diaphragm and that of walls in resisting lateral loads depends on the efficiency of the fastening systems and connection details used to connect individual panels and assemblies. Consequently, they require detailed attention by designers. In addition, damages and failures in X-Lam buildings during a seismic event are localized in connections; thus, structural repairs after an earthquake are relatively easy and cost effective.

When structural members are attached with fasteners or some other types of metal hardware, such joints are referred to as “mechanical connections”. Currently, there is a wide variety of mechanical fasteners and many different types of joint details that can be used for panel to panel connections in X-Lam assemblies or to connect X-Lam panels to other wood-based, concrete or steel elements in hybrid construction. A combination of metal hold-downs, angle brackets and self-tapping screws are typically recommended by the X-Lam manufacturers for connecting the cross-lam panels.

Metal brackets, hold-downs, plates and straps are used to transfer forces from walls to floors, from one level to another level, and to foundations. Hold-downs are mainly used in the corners of wall segments and close to door opening, to resist overturning forces that result from an earthquake or wind. On the other hand, the main role of L-shaped metal brackets is to resist shear forces in wall panels caused by wind or

a seismic event. Nails with specific surface features such as grooves or helically threaded nails are mostly used with perforated metal plates and brackets and installed on the surface of the panel.

Long self-tapping screws are typically recommended by X-Lam manufacturers due to their ease of installation along with high lateral and withdrawal capacity, which make these fasteners popular because they can take combined axial and lateral loads.

Bolts and dowels are very common in heavy timber construction. They can also be used in the assembly of X-Lam panels, especially for lateral loading. If installed in the narrow face, care must be taken during the design, especially in X-Lam panels with unglued edges between the individual planks in a layer. This could eventually compromise the lateral resistance since there is a potential that such fasteners are driven in the gaps.

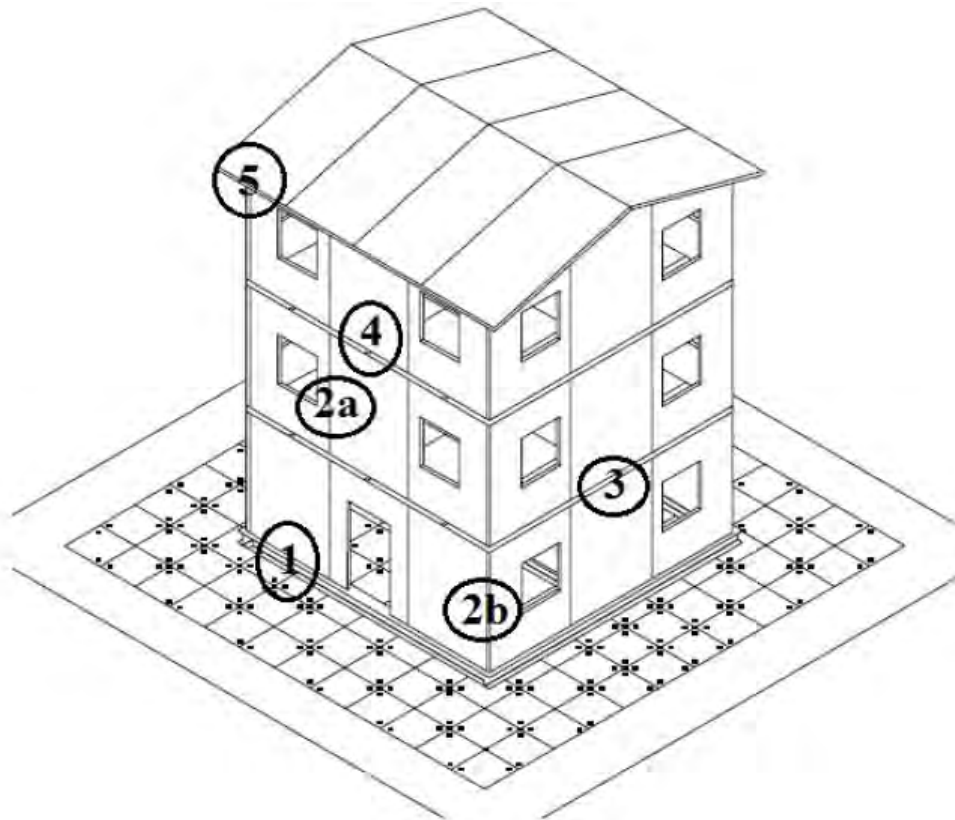


Figure 2.6 Typical three-storey X-Lam building showing various connections between the X-Lam panels

However, there are other types of traditional and innovative fasteners and fastening systems that can be used efficiently in X-Lam assemblies. The choice of the type of connection to use depends largely on the type of assemblies to be connected, panel configurations, and the type of structural system used in the building. With these mechanical connections, several possibilities for assembling X-Lam panels are possible, as shown in Figure 2.6.

1. Wall-to-foundation connections

Several fastening systems are available for connecting X-Lam wall panels to steel beams or to concrete foundations with concrete footing, which are most common for the ground storeys in X-Lam buildings.

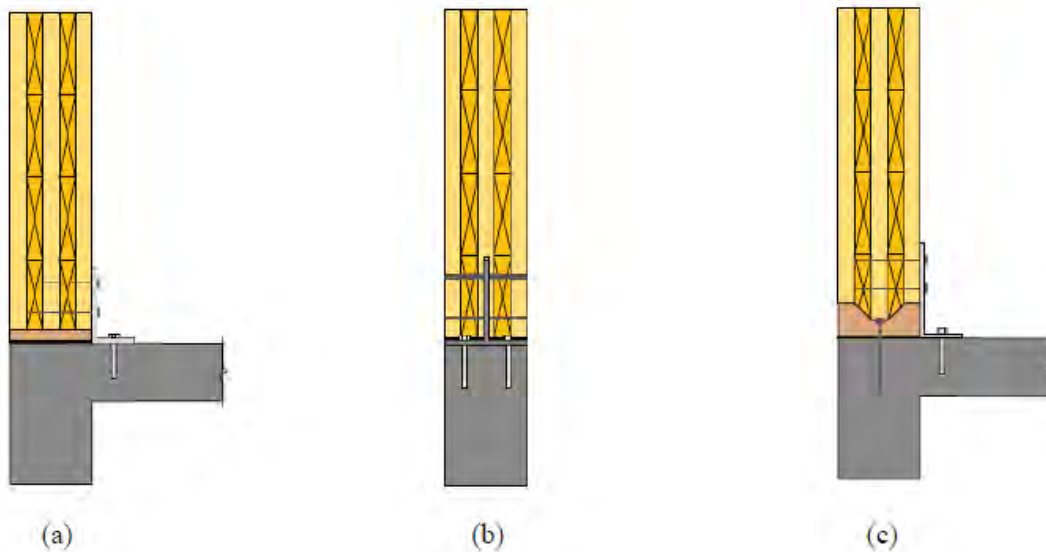


Figure 2.7 Typical wall-to-foundation X-Lam connections: (a) Connection with an exposed metal plate; (b) Connection with a concealed connector; (c) Connection with a wooden profile (FPInnovations, 2013)

- **Visible or exposed metal plates**

Exterior metal plates and brackets are commonly used in such applications as there is a variety of such metal connectors readily available on the market and due to its simple installation. Lag screws or powder-actuated fasteners can be used to connect the metal plate to the concrete footing or slab, while nails, lag screws or self-tapping screws are

used to connect the plate to the X-Lam panel. Exposed metal plates and fasteners need to be protected against corrosive exterior environments. Galvanized or stainless steel should be used in such cases. Direct contact between the concrete foundation and X-Lam panel should be avoided in all cases. Connection details should be designed to prevent potential moisture penetration between the metal plates and the X-Lam wall as water may get trapped and cause potential decay of the wood.

- **Concealed connectors**

For better fire resistance and improved aesthetics, designers sometimes prefer concealed connection systems. This can be achieved with hidden metal plates. However, some CNC machining work is required to produce the grooves in the X-Lam panel to conceal the metal plates. Tight dowels or bolts can be used to attach the plates to the X-Lam panel. In addition, some innovative types of fasteners that can be drilled through metal and wood or other types of screws that can penetrate through both materials can also be used for this purpose.

- **Wooden Profiles**

Wooden profiles, which are fabricated from high density and stable materials such as engineered wood products or high density hardwood, are commonly used for connecting structural insulated panels (SIP) and other types of prefabricated wood-framed walls. The major advantage of this system is the ease of assembly. The wooden profiles are typically attached to X-Lam panels with wood screws or self-tapping screws and are often used in combination with metal plates or brackets to improve the lateral load resistance. Wooden profiles can also be used for wall-to-wall or floor-to-wall connections.

2. Wall-to-wall connections

2a) Parallel wall panel connection

This connection type is used to connect panels along their longitudinal edges. The parallel wall-wall panel connection facilitates the transfer of in-plane forces (shear) and

out-of-plane forces (bending) through the wall assembly. Several connections details are possible.

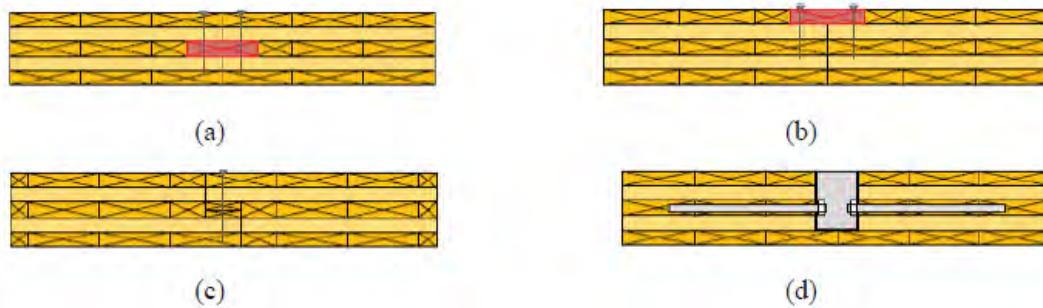


Figure 2.8 Typical parallel wall-to-wall X-Lam connections: (a) Connection with an internal spline; (b) Connection with a surface spline; (c) Connection with a half-lapped joint; (d) Tube connection system (FPInnovations, 2013)

- **Internal splines or strips**

For formation of this connection type, single or double wooden splines (strips) made of structural composite lumber, such as laminated veneer lumber (LVL), plywood or thin X-Lam, are used. Connection between the spline or splines and the panel edges can be established using self-tapping screws, wood screws or nails. The advantage of this detail is that it provides a double shear connection and resistance to out-of-plane loading. However, special attention is required due to necessity of accurate profiling for the fitting of different parts on a construction site.

- **Surface splines or strips**

This fairly simple connection detail can be established quickly on site but it only provides single shear connection. Since two sets of screws are used, which results in doubling the number of shear planes resisting the load, a better resistance can be achieved using this detail. Panel edges are profiled from one side for a single surface spline or from both sides for a double surface spline. Similarly as in the case of internal splines, structural composite lumber elements are used for strips. Traditional fasteners such as nails, self-tapping screws, wood screws and lag screws can be used for making the connection on site. In case of double surface spline connection, the strength and stiffness of the connection can be increased. If SCL (structural composite lumber) is

used as the spline, the joint can be designed to resist moment for out-of-plane loading (Augustin, 2008). Structural adhesives could be used to enhance the strength and stiffness.

- **Half-lapped joint**

In this connection type, long self-tapping screws are usually used to connect the panel edges. The joint can carry normal and transverse loads but it is not considered to be a moment resisting connection (Augustin, 2008). This connection detail is considered as very simple, so it facilitates quick assembly of X-Lam elements. However, there is a risk of splitting of the cross section due to concentration of tension perpendicular to grain stresses in the notched area. This is particularly emphasized for cases where uneven loading on the floor elements occur (Augustin, 2008).

- **Tube connection system**

Tube connection system incorporates a profiled steel tube with holes in the X-Lam panel. Panel elements are delivered on site with glued-in or screwed rods driven in the plane of the two panels which are supposed to be connected. The tube connector is inserted at certain locations along the edges of the panels where the metal tubes are to be placed. The system is tightened on site using metal nuts. Usually no edge profiling along the panel is needed as it relies principally on the pull-out resistance of the screwed or glued-in rods (Traetta, 2007).

2b) Perpendicular wall panel connection

This Section presents connection details for connecting wall panels to wall panels positioned at right angles (transverse direction). Such connection details include interior partition walls to exterior walls or just exterior corner walls. Several systems have been adopted to establish connection between perpendicular walls.

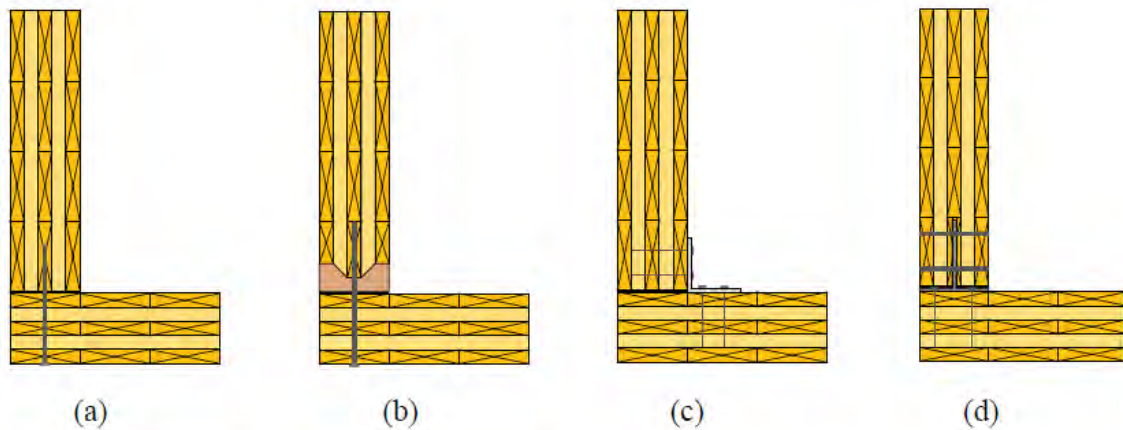


Figure 2.9 Typical perpendicular wall-to-wall X-Lam connections: (a) Connection with self-tapping screws; (b) Connection with a wooden profile; (c) Connection with a metal bracket; (d) Connection with a concealed metal plate (FPInnovations, 2013)

- **Self-tapping screws**

This is the simplest form of connecting X-Lam wall panels together. There are some concerns related to this form of connection due to the fact that the screws are driven in the narrow side of panels, in particular if screws are installed in the end grain of the cross layers. Self-tapping screws can be driven straight into the X-Lam panel or at an angle to avoid direct installation of screws in the narrow side of the panel.

- **Wooden profiles**

Concealed wooden profiles or keys can also be used in a similar way, with self-tapping screws or traditional wood screws. The advantage of this system over the direct use of self-tapping screws is the possibility of enhancing the connection resistance by driving more wood screws to connect the profiled panel to the central wood profile which is in turn screwed to the transverse wall.

- **Metal brackets**

Another simple form of connecting walls in the transverse direction is the use of metal brackets with screws or nails. This connection system is one of the simplest and most efficient types of connection in terms of strength resulting from fastening in the

direction perpendicular to the plane of the panels or recessed. However, adding protective membrane (i.e., gypsum board) for improved fire resistance is required.

- **Concealed metal plates**

As previously discussed, while it has considerable advantages over exposed plates and brackets, especially when it comes to fire resistance, this system requires precise profiling at the plant using CNC machining technology. Self-drilling dowels that can penetrate through wood and steel can also be used. Metal plate thickness ranges from 6 mm up to 12 mm.

3. Wall-to-floor connections

Several possibilities exist when it comes to connecting walls to the floors above, depending on the form of structural systems (i.e., platform or balloon), on the availability of fasteners and the degree of prefabrication.

3a) Platform construction

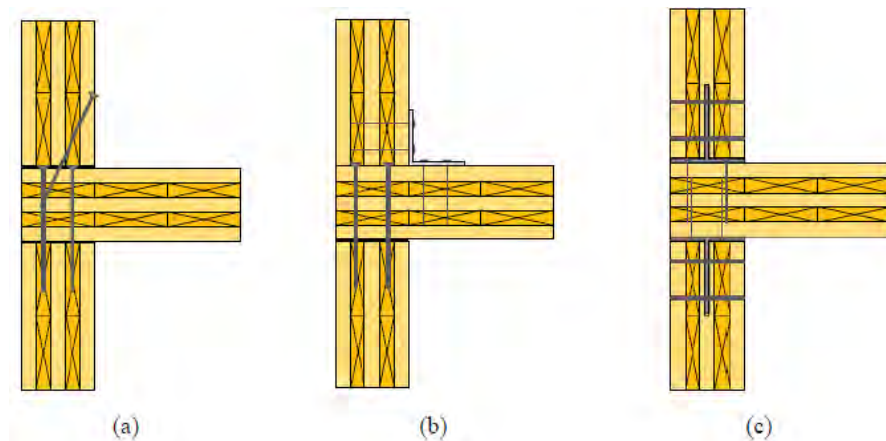


Figure 2.10 Typical wall-to-floor X-Lam connections: (a) Connection with self-tapping screws; (b) Connection with a metal bracket; (c) Connection with concealed metal plates (FPInnovations, 2013)

- **Self-tapping screws**

The simplest method for connecting a floor or a roof to walls below is to use long self-tapping screws driven from the X-Lam floor directly into the narrow side of the wall edge. Self-tapping screws can also be driven at an angle to maximize the fastening capacity in the panel edge. The same principle could be applied for connecting walls above to floors below, where self-tapping screws are driven at an angle in the wall near the junction with the floor. However, this type of connection has relatively low seismic capacity in terms of strength and stiffness (Popovski, 2010).

- **Metal brackets**

Metal L-shaped brackets are commonly used to connect floors to walls above and below to transfer lateral loads from diaphragm to shear walls. Nails or wood screws can be used to attach the metal brackets to the X-Lam panels. They are also used for connecting roofs to walls.

- **Concealed metal plates**

As discussed before, while concealed metal plates have considerable advantages over exposed plates and brackets (especially fire resistance), the system requires precise profiling at the plant using CNC machining technology.

3b) Balloon construction

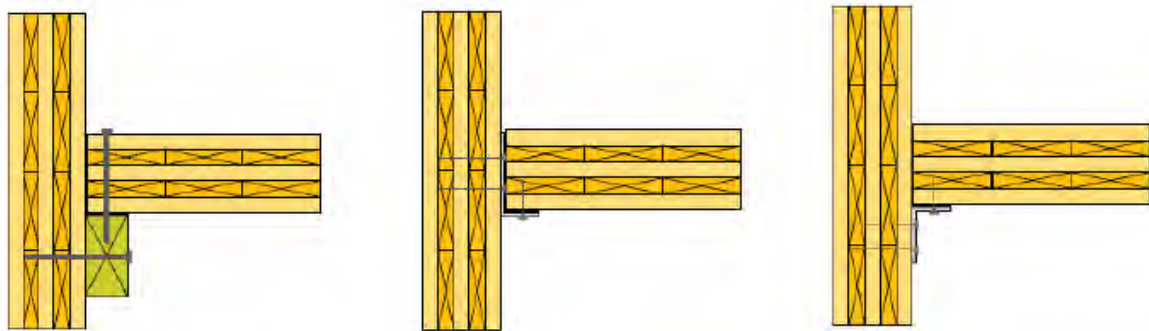


Figure 2.11 Typical wall-to-floor X-Lam connections in balloon construction (FPInnovations, 2013)

In Europe, the most common type of structural form in X-Lam construction is the platform type of system due to its simplicity in design and erection. However, in non-residential construction, including industrial buildings, it is common to use tall walls with an intermediate floor between the main floors of a building. So called “mezzanine floor” is often located between the ground floor and the first floor. However, it is not unusual to have a mezzanine in the upper floors of a building. Several attachment options to connect X-Lam floor to a continuous X-Lam tall wall exist for such applications. The simplest attachment detail includes the use of a wooden ledger (made of structural composite lumber) to provide a continuous bearing support to the X-Lam floor panels. Another type of attachment is established with the use of metal brackets. Attachment of wooden ledger or metal brackets to the X-Lam wall and floor panels is established through the use of screws, lag screws or nails. However, out-of-plane bending due to wind suction could be an issue with this type of detail and designers need to take that into account.

4. Floor-to-floor connections

When the connection is used in floor assemblies acting as diaphragms, the connection must be capable of transferring in-plane diaphragm forces in principle, and maintain the integrity of the diaphragms. Connection details used in floor-to-floor panel connection are equal to parallel wall-to-wall panel connection types, described earlier in this Chapter.

5. Wall-to-roof connections

For sloped or flat roof systems, connections similar to those used for attaching floors to walls is used. Screws and metal brackets are the most commonly used fastening systems in this application.

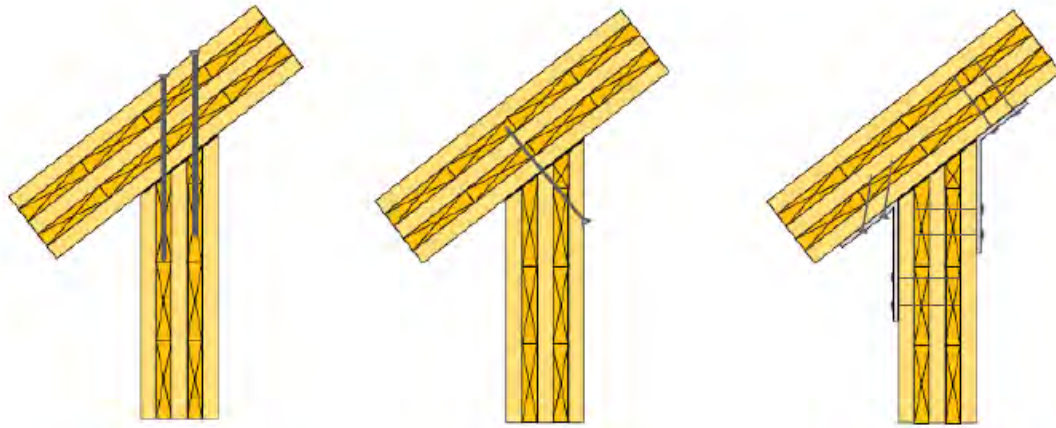


Figure 2.12 Typical wall-to-roof X-Lam connections (FPInnovations, 2013)

Innovative types of connection systems can also be used, including mechanical and carpentry connection systems. Some interesting innovative connection systems are finding their way to the X-Lam construction market, mostly facilitated and enabled by CNC technology. For example, glued-in rods can be used for connections under high longitudinal and transverse loads. HBV-Shear Connectors, a proprietary product from Germany, can also be used to create composite floors with structural concrete over X-Lam panels (Bathon & Bletz, 2006). Further, KNAPP system (Knapp) and Idefix connectors (Sihga) are relatively new innovative connection systems on the market. Due to the relatively recent introduction of X-Lam technology into the construction market, it is expected that even more new connection types will be developed over time.



Figure 2.13 Sihga Idefix innovative connection system (Sihga)

2.7.1 Connections behaviour

About the kinematic behaviour of the connections, it can be assumed that:

1. the shear connections (angle brackets, shear screws,...) do not work in tension;
2. the tension connections (hold-down) do not work in shear.

These two hypotheses are, nowadays, widely used in the calculation of X-Lam buildings and they are assumed to be valid in this thesis. However, if on one hand they allow to greatly simplify the calculations, on the other hand they do not take into account the actual failure behaviour of the panels. The behaviour of an X-Lam panel subjected to horizontal forces is a combination of horizontal sliding and rigid rotation (rocking); consequently, all connections at the base are subjected to a combination of horizontal forces, or shear forces, and lifting. For further information about the behaviour of an X-Lam panel subjected to horizontal forces, see the thesis “*Una procedura numerica per il progetto di edifici in X-Lam*” by Massimiliano Zecchetto.

Relatively to the first assumption, different theoretical models have been proposed in order to take into account the actual presence of angle brackets in the rocking effect (Gavric and Popovski, 2014). In this case, the interaction between shear and tension, to which the angle brackets are subjected, must also be considered. Finally, it results necessary to define a domain of resistance in the plane N - V.

Regarding the second assumption, the hypothesis is fully confirmed by experimental research on X-Lam walls (Popovski, 2010) and on individual connecting elements (Gavric, Fragiaco and Ceccotti, 2014).

2.7.2 Connections stiffness

Regardless of the behaviour of the connections, shear or tension, the code UNI-EN 1995-1-1:2009 provides the relations for the calculation of the slip modulus k_{ser} per shear plane per fastener, which are shown in Table 2.6.

Table 2.6 Values of K_{ser} for fasteners and connectors in N/mm in timber-to-timber and wood-based panel-to-timber connections, according to UNI-EN 1995-1-1: 2009 (the density ρ_m is expressed in kg/m³ and the diameter is expressed in mm)

Fastener type	K_{ser}
Dowels Bolts with or without clearance ^a Screws Nails (with pre-drilling)	$\rho_m^{1,5} d/23$
Nails (without pre-drilling)	$\rho_m^{1,5} d^{0,8}/30$
Staples	$\rho_m^{1,5} d^{0,8}/80$

The slip modulus k_{ser} for fasteners in timber-to-steel connections is equal to two times the slip modulus in timber-to-timber connections.

Once identified the stiffness per shear plane per fastener, the stiffness of the brackets or distributed nailing and hold-down can be calculated.

The stiffness of the brackets is obtained by simply multiplying k_{ser} by the total number of nails of the connection (meaning the number of nails if only one bracket or a distributed nailing is disposed, the number of nails multiplied by the number of brackets if more than one bracket are disposed).

Similarly, the stiffness of the hold-down can be calculated by multiplying k_{ser} by the total number of nails of the connection (meaning the product between the number of nails of each hold-down and the number of hold-down). However, in this case, the stiffness inserted in the model does not coincide with the stiffness value suggested by the code. This comes from the difference in behaviour, under the action of horizontal forces, of a single modelled panel compared with the same in a real situation (see Section 4.3).

2.7.3 Connections resistance

The design resistance of a connection R_d (load-carrying capacity) can be calculated as:

$$R_d = k_{mod} \frac{R_k}{\gamma_M}$$

where R_k is the characteristic value of load-carrying capacity, γ_M is the partial factor for a material property (according to Table 2.3), k_{mod} is the correction factor taking into account the effect of the load duration and moisture content of the timber (according to Table 2.2).

Assuming that the characteristic load-carrying capacity of a connection is governed by the number of fasteners, then:

$$R_k = \eta_{eff} \cdot F_{V,Rk}$$

being $F_{V,Rk}$ the characteristic load-carrying capacity of a single fastener and η_{eff} the effective number of fasteners of the connection itself. This assumption leads to impose a hierarchy of resistance within the single connection; the relationship considered is only valid in case of failure of the connection on the fasteners and not on the steel plate. The resistance of the connection would actually be equal to:

$$R_k = \min \begin{cases} R_{plate} = N_{t,Rk} \\ R_{fast} = \eta_{eff} \cdot F_{V,Rk} \end{cases}$$

being $N_{t,Rk}$ the characteristic value of the tensile strength for steel plates with holes, as defined in the UNI-EN 1993-1-1: 2005. This assumption, in fact, implies further verifications after the post-process to check that the actual failure mode of each connection is the one assumed. This verification will imply the following main problems, whose checks will be not deepened in this thesis:

- statistical distribution of all the resistances;
- different safety factors for the resistances $F_{V,Rk}$ and $N_{t,Rk}$;
- different resistance values for a single fastener.

The characteristic load-carrying capacity $F_{V,Rk}$ of a single fastener can be calculated, according to UNI-EN 1995-1-1: 2009, as the minimum of all the resistances associated to the possible modes of failure of a single fastener. The code differentiates the timber-to-timber and panel-to-timber connections from the steel-to-timber ones

(Figure 2.14). For each of these families both possible failure modes and relative resistance values, referred to single fastener, are shown.

The characteristic load-carrying capacity for nails, staples, bolts, dowels and screws per shear plane per fastener, in case of timber and panel connections, can be calculated in different ways, for fasteners in single and double shear.

For fasteners in single shear it should be taken as the minimum value found from the following expressions (with reference to Figure 2.14a):

$$F_{v,Rk} = \min \left\{ \begin{array}{l} f_{h,1,k} t_1 d \quad (a) \\ f_{h,2,k} t_2 d \quad (b) \\ \frac{f_{h,1,k} t_1 d}{1 + \beta} \left[\sqrt{\beta + 2\beta^2 \left[1 + \frac{t_2}{t_1} + \left(\frac{t_2}{t_1} \right)^2 \right]} + \beta^3 \left(\frac{t_2}{t_1} \right)^2 - \beta \left(1 + \frac{t_2}{t_1} \right) \right] + \frac{F_{ax,Rk}}{4} \quad (c) \\ 1,05 \frac{f_{h,1,k} t_1 d}{2 + \beta} \left[\sqrt{2\beta(1 + \beta) + \frac{4\beta(2 + \beta)M_{y,Rk}}{f_{h,1,k} d t_1^2}} - \beta \right] + \frac{F_{ax,Rk}}{4} \quad (d) \\ 1,05 \frac{f_{h,2,k} t_1 d}{2 + \beta} \left[\sqrt{2\beta(1 + \beta) + \frac{4\beta(2 + \beta)M_{y,Rk}}{f_{h,1,k} d t_2^2}} - \beta \right] + \frac{F_{ax,Rk}}{4} \quad (e) \\ 1,15 \sqrt{\frac{2\beta}{1 + \beta}} \sqrt{2M_{y,Rk} f_{h,1,k} d} + \frac{F_{ax,Rk}}{4} \quad (f) \end{array} \right.$$

For fasteners in double shear it should be taken as the minimum value found from the following expressions (with reference to the Figure 2.14b):

$$F_{v,Rk} = \min \left\{ \begin{array}{l} f_{h,1,k} t_1 d \quad (g) \\ 0,5 f_{h,2,k} t_2 d \quad (h) \\ 1,05 \frac{f_{h,1,k} t_1 d}{2 + \beta} \left[\sqrt{2\beta(1 + \beta) + \frac{4\beta(2 + \beta)M_{y,Rk}}{f_{h,1,k} d t_1^2}} - \beta \right] + \frac{F_{ax,Rk}}{4} \quad (j) \\ 1,15 \sqrt{\frac{2\beta}{1 + \beta}} \sqrt{2M_{y,Rk} f_{h,1,k} d} + \frac{F_{ax,Rk}}{4} \quad (k) \end{array} \right.$$

where:

$F_{V,Rk}$ is the characteristic load-carrying capacity per shear plane per fastener;

t_i is the timber or board thickness or penetration depth, with i either 1 or 2;

$f_{h,i,k}$ is the characteristic embedment strength in timber member i ;

d is the fastener diameter;

$M_{y,Rk}$ is the characteristic fastener yield moment;

β is the ratio between the embedment strength of the members;

$F_{ax,Rk}$ is the characteristic axial withdrawal capacity of the fastener.

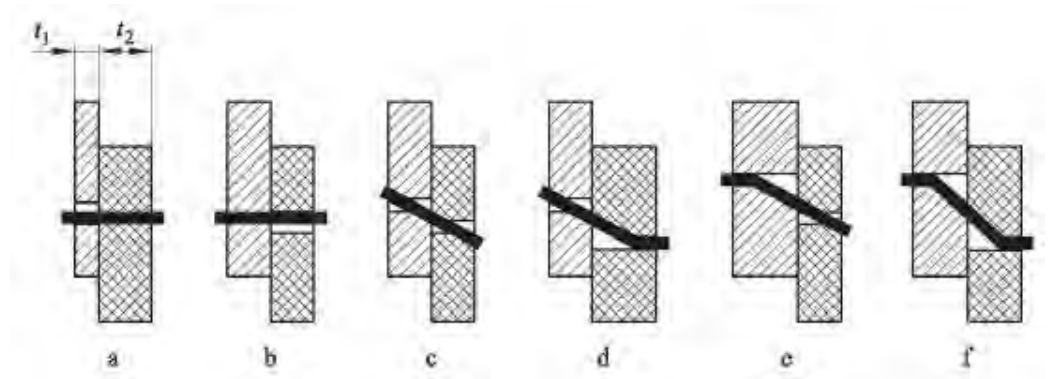


Figure 2.14(a) Failure modes for timber and panel connections with single shear (UNI-EN 1995-1-1: 2009)

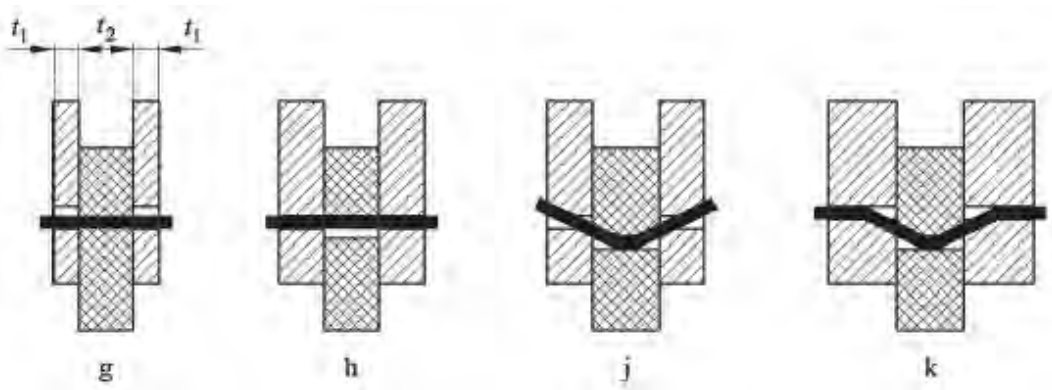


Figure 2.14(b) Failure modes for timber and panel connections with double shear (UNI-EN 1995-1-1: 2009)

The characteristic load-carrying capacity of a steel-to-timber connection depends on the thickness of steel plates. Steel plates of thickness less than or equal to $0.5d$ are classified as thin plates and steel plates of thickness greater than or equal to d , with the tolerance on hole diameters being less than $0.1d$, are classified as thick plates. The characteristic load-carrying capacity of connections with steel plate thickness between a thin and a thick plate should be calculated by linear interpolation between the limiting thin and thick plate values.

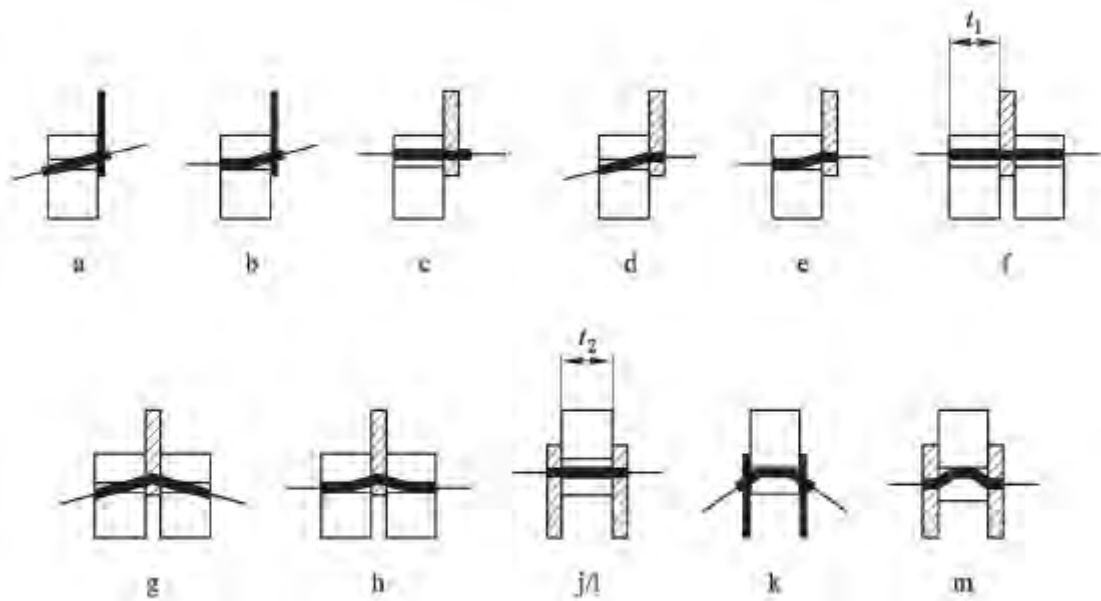


Figure 2.14(c) Failure modes for steel-to-timber connections (UNI-EN 1995-1-1: 2009)

The characteristic load-carrying capacity for nails, bolts, dowels and screws per shear plane per fastener, in case of steel-to-timber connections, should be taken as the minimum value found from the following expressions (with reference to the Figure 2.14c).

- For a thin steel plate in single shear:

$$F_{v,Rk} = \min \begin{cases} 0,4 f_{h,k} t_1 d & \text{(a)} \\ 1,15\sqrt{2 M_{y,Rk} f_{h,k} d} + \frac{F_{ax,Rk}}{4} & \text{(b)} \end{cases}$$

- For a thick steel plate in single shear:

$$F_{v,Rk} = \min \begin{cases} f_{h,k} t_1 d & \text{(c)} \\ f_{h,k} t_1 d \left[\sqrt{2 + \frac{4M_{y,Rk}}{f_{h,k} d t_1^2}} - 1 \right] + \frac{F_{ax,Rk}}{4} & \text{(d)} \\ 2, 3\sqrt{2 M_{y,Rk} f_{h,k} d} + \frac{F_{ax,Rk}}{4} & \text{(e)} \end{cases}$$

- For a steel plate of any thickness as the central member of a double shear connection:

$$F_{v,Rk} = \min \begin{cases} f_{h,1,k} t_1 d & \text{(f)} \\ f_{h,k} t_1 d \left[\sqrt{2 + \frac{4M_{y,Rk}}{f_{h,k} d t_1^2}} - 1 \right] + \frac{F_{ax,Rk}}{4} & \text{(g)} \\ 2, 3\sqrt{2 M_{y,Rk} f_{h,1,k} d} + \frac{F_{ax,Rk}}{4} & \text{(h)} \end{cases}$$

- For thin steel plates as the outer members of a double shear connection:

$$F_{v,Rk} = \min \begin{cases} 0, 5 f_{h,2,k} t_2 d & \text{(j)} \\ 1, 15\sqrt{2 M_{y,Rk} f_{h,2,k} d} + \frac{F_{ax,Rk}}{4} & \text{(k)} \end{cases}$$

- For thick steel plates as the outer members of a double shear connection:

$$F_{v,Rk} = \min \begin{cases} f_{h,2,k} t_2 d \\ 2, 3\sqrt{M_{y,Rk} f_{h,2,k} d} + \frac{F_{ax,Rk}}{4} \end{cases} \quad \text{(m)}$$

where:

$F_{v,Rk}$ is the characteristic load-carrying capacity per shear plane per fastener;

$f_{h,k}$ is the characteristic embedment strength in the timber member;

t_1 is the smaller of the thickness of the timber side member or the penetration depth;

t_2 is the thickness of the timber middle member;

d is the fastener diameter;

$M_{y,Rk}$ is the characteristic fastener yield moment;

$F_{ax,Rk}$ is the characteristic withdrawal capacity of the fastener.

In all expressions of $F_{V,Rk}$ above, the first term on the right hand side is the load-carrying capacity according to the Johansen yield theory, while the second term $F_{ax,Rk}/4$ is the contribution from the rope effect. The contribution to the load-carrying capacity due to the rope effect should be limited to following percentages of the Johansen part:

- Round nails: 15%;
- Square and grooved nails: 25%;
- Other nails: 50%;
- Screws: 100%;
- Bolts: 25%;
- Dowels: 0%.

2.8 X-Lam structural applications

This Section presents an introduction to the X-Lam construction systems and their applications. There are several ways to design and construct X-Lam buildings. They all differ in the way the load-carrying elements (panels) are arranged, the way the panels are connected together and by the type of wood and non-wood based materials used. The most common forms of X-Lam construction systems are platform construction and so called balloon construction (FPInnovations, 2013).

Platform construction in X-Lam technology is a system where the floor panels rest directly on top of wall panels, therefore forming a platform for subsequent floors. This is the most commonly used type of structural system for X-Lam assemblies for multi-storey buildings. This includes buildings constructed with X-Lam panels only or combining the panels with other types of wood-based products, for example glulam. There are several advantages to this system, such as quicker erection of upper storeys, possible application of simple connection systems and well-defined load path.

Balloon construction is a type of structural system where the walls continue for a few storeys with intermediate floor assemblies attached to those walls. Due to the

limitations in the length of the X-Lam panels, this system is often used in low-rise, commercial or industrial buildings; connections are usually more complex in this form of construction. Balloon construction is generally less common compared to platform construction.

X-Lam solid wood panels are used both as load-bearing, reinforcing elements and non-load-bearing elements. Areas of application:

- houses and apartment buildings
- multi-storey residential buildings
- public buildings
- hotels and restaurants
- schools and kindergartens
- offices and administrative buildings
- event halls
- industrial and commercial buildings
- reconstructions, extensions and upgrades
- building retrofits
- bridges

Numerous buildings using X-Lam panels have already been erected around the world, starting in Europe, and recently some projects were realized in North America and in Australia. In Europe, the tallest X-Lam structure to date is the 9-storey Stadthaus residential building in London, which includes eight storeys of X-Lam over one storey of concrete. At the time of the erection, in 2009, this building was the tallest wooden residential building in the world. Short erection time, environmental benefits and cost efficiency of the building illustrated how X-Lam can be a competitive system in the marketplace (Yates et al., 2008). In 2011, another multi-storey residential building was constructed in London, named Bridport House. It is consisted of two joined blocks, one eight storeys high and the other five storeys high, both entirely built with X-Lam technology, including the ground floor, which is traditionally made of concrete (Stora Enso). Total erection time of the building was only 12 weeks. X-Lam construction system is also gaining popularity for educational buildings such as the Norwich Open

Academy, also in the UK (KLH). In Austria, numerous numbers of hotels, single-family and multi-family X-Lam buildings were realized in the last decade (KLH). A project of four residential X-Lam buildings, each 9 storeys tall, started in 2012 in Milano, Italy (Stora Enso). In Växjö, Sweden, four 8-storey residential X-Lam building were built in 2008. For each floor, four construction days were needed (Martinson). Recent initiatives of introducing the X-Lam technology overseas, namely in North America and Australia, resulted in realisation of several interesting projects. In Melbourne, Australia, a 10-storey X-Lam building has been erected, making it the tallest residential wooden building in the world (KLH).

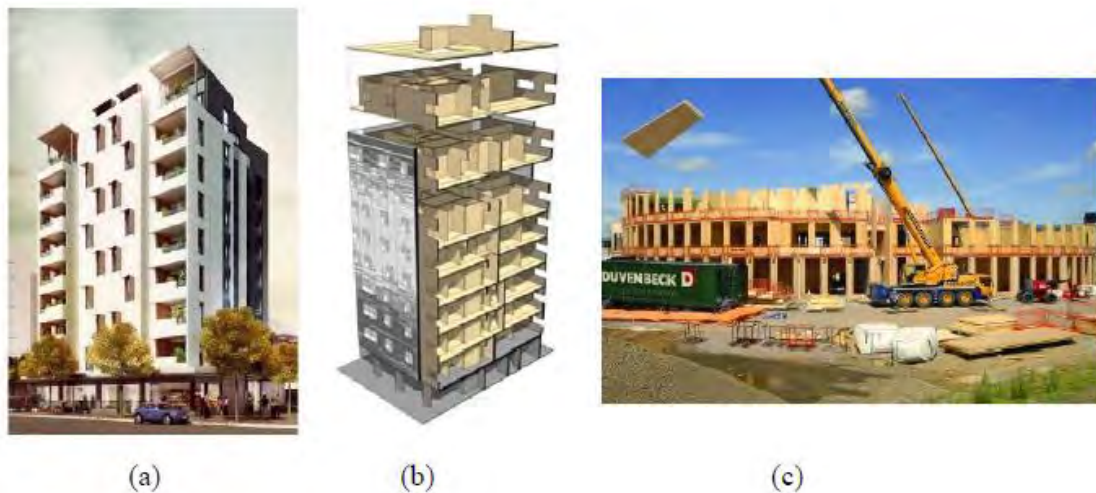


Figure 2.15 Residential and non-residential X-Lam projects: (a) 10-storey Forté in Melbourne; (b) 9-storey Stadthaus in London; (c) Open Academy in Norwich (KLH)

A four-storey building on the University of British Columbia campus, Vancouver, Canada, was the first North American commercial application in X-Lam technology. The building was built using a combination of massive timber systems including X-Lam, composite laminated strand lumber with concrete floors, and glulam heavy timber braced frames (Wood Works). A prototype of a wind turbine was currently built from X-Lam panels in Hannover, Germany; the structure reaches 100 m in height (Timber Tower).

CHAPTER 3

GENERAL ABOUT GiD, KRATOS, TCL AND PYTHON

This Chapter provides an overview of general information about the processor, the solver and the programming languages used for this thesis: GiD, Kratos, Tcl and Python. Thus, only a brief description rather than a detailed explanation of each one of them will be presented, in order to provide the reader an idea of what they are and how they work.

3.1 GiD processor



GiD is a universal and adaptive pre- and post-processor for numerical simulations in science and engineering. It has been designed to cover all the common needs in the numerical simulations field from pre- to post-processing: geometrical modelling, effective definition of analysis data, meshing, data transfer to analysis software, as well as visualization of numerical results.

3.1.1 Interaction of GiD with the calculating module

GiD pre-process makes a discretization of the object under study and generates a mesh of elements, to each one of whom is assigned a material and some conditions. This pre-processing information in GiD (mesh, materials, and conditions) enables the

calculating module to generate results. Finally, the results generated by the calculating module will be read and visualized in GiD post-process. GiD must adapt these data to deal with them. Materials, boundary and/or load conditions, and general problem data must be defined by the user.

GiD configuration is accomplished through text formatted files. The following files are required:

- **.prb**: configuration of the general parameter (not associated to entities).
- **.mat**: configuration of materials and their properties.
- **.cnd**: configuration of the conditions imposed on the calculation.
- **.bas** (template file): the file for configuring the format of the interchange file that mediates between GiD data and the calculating module. The file for interchanging the data exported by GiD has the extension **.dat**. This file stores the geometric and physical data of the problem.
- **.bat**: the file that can be executed called from GiD. This file initiates the calculating module.

The calculating module solves the equations in the problem and saves the results in the results file; this module may be programmed.

GiD post-process reads the following files generated by the calculating module:

- **project_name.post.res**: results file. Each element of the mesh corresponds to a value.
- **project_name.post.msh**: file containing the post-process mesh. If this file does not exist, GiD uses the pre-process mesh also for post-process.

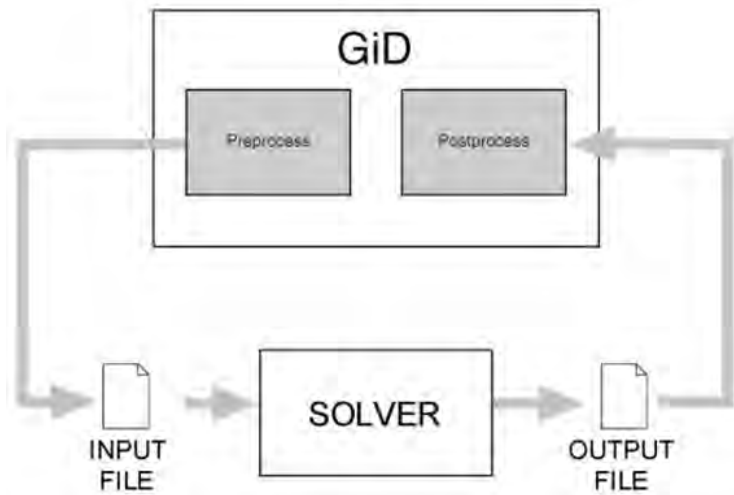


Figure 3.1 Diagram of GiD workflow (GiD User Manual)

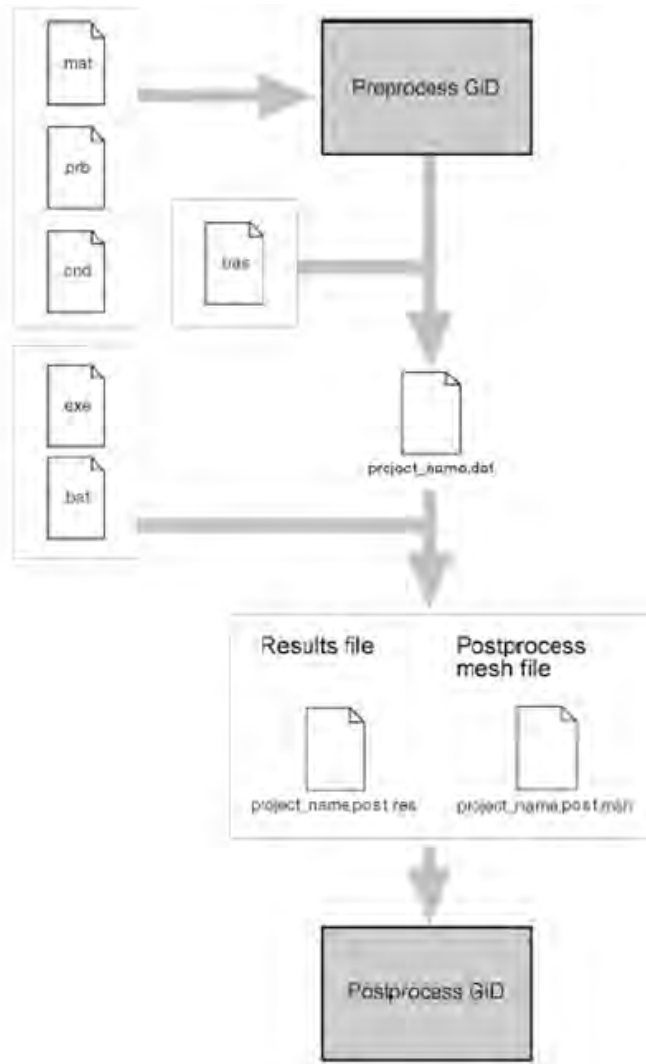


Figure 3.2 Diagram depicting the files system (GiD User Manual)

3.1.2 GiD Pre-process

GiD is a CAD system that features the widely used nurbs surfaces for the geometry definition. Typical geometrical operations can be used as transformations (translations, rotations, etc.), boolean operations in surfaces and volumes; a complete set of tools are provided for quick geometry definition.

GiD allows for the generation of large meshes (for linear and quadratic elements) in a fast and efficient manner using several in-house meshers, both for surfaces and volumes, following different structured type criteria:

- structured mesher including triangular, quadrilateral, hexahedral, prism and tetrahedral meshes;
- unstructured meshes are automatically generated based on quality and spacing criteria defined by the user (or using a background mesh for defining a certain size distribution). Several element types can be generated: triangular, quadrilateral, circles, spheres and tetrahedral.

GiD also can generate 2D and 3D anisotropic meshes (useful for boundary layer). Several mesh editing tools like edge collapsing, elements splitting, smoothing, etc. allow the user to have the total control of any kind of mesh.

Simple assignment of any kind of data to the geometry and/or the mesh (boundary conditions, material properties, etc.) is available. This information can be sent to the solver with other analysis data, which is easily included because of GiD customization.

3.1.3 GiD Post-process

All the widely used types of visualization for the numerical results coming from simulations that are present in GiD, such as contour fill and contour lines, vector plots, isosurfaces, beam diagrams, stream lines and ribbons, surface extrusions, deformations,

etc.. Each visualization type has several options such as showing the contour fill of a result over an isosurface of another result.

GiD also offers the possibility of visualizing the results on several meshes for adaptive solutions, combining different visualization styles and results, and creation of animated sequences.

All the typical graph types can be done in GiD, like point evolution, which shows the evolution of a result of a point across all time steps of an analysis, line graph, boundary graph and point analysis, in which a result can be plotted against another one for a point and, optionally, for all time steps.

3.2 Kratos solver



Kratos is a framework for the implementation of numerical methods for the solution of engineering problems. It is written in C++ language and is designed to allow for collaborative development by large teams of researchers focusing on modularity as well as on performance. The Kratos features a "core" and "applications" approach where "standard tools" (databases, linear algebra, search structures, etc...) come as a part of the core and are available as building blocks in the development of "applications" which focus on the solution of the problems of interest. Its ultimate goal is to simplify the development of new numerical methods.

Kratos provides several tools for easy implementation of finite element applications and a common platform providing effortless interaction between them. It has an innovative variable base interface designed to be used at different levels of abstraction and implemented to be very clear and extendible. It also provides an efficient yet flexible data structure which can be used to store any type of data in a type-

safe manner. The Python scripting language is used to define the main procedure of Kratos which significantly improves the flexibility of the framework in time of use.

The kernel and application approach is used to reduce the possible conflicts arising between developers of different fields. Also layers are designed to reflect the working space of different people, considering their programming knowledge.

Kratos is parallelized for Shared Memory Machines (SMMs) and Distributed Memory Machines (DMMs). In the same way it provides tools for its applications to adapt easily their algorithms to these architectures; its scalability has been verified up to thousands of cores.

The Kratos structure, due to its multi-disciplinary nature, has to support the wide variety of algorithms involved in different areas. That's the principal reason which explains the variety of people, mostly engineers, composing the Kratos Community.

Some potential users of Kratos are:

- **Finite Element Developers:** These developers are considered to be more expert in FEM, from the physical and mathematical points of view, than C++ programming. For this reason, Kratos provides their requirements without involving them in advanced programming concepts.
- **Application Developers:** These users are less interested in finite element programming and their programming knowledge may vary from very expert to higher than basic. They may use not only Kratos itself but also any other application provided by finite element developers, or other application developers. Developers of optimization programs or design tools are the typical users of this kind.
- **Package Users Engineers:** designers are other users of Kratos. They use the complete package of Kratos and its applications to model and solve their problem without getting involved in internal programming of this package. For these users Kratos has to provide a flexible external interface to enable them use different features of Kratos without changing its implementation.

3.2.1 Kratos advantages

Kratos is multi-physic. One of the main topics in engineering nowadays is the combination of different analysis (thermal, fluid dynamic, structural) with optimising methods in one global software package with just one user interface and, even more, the possibility to extend the implemented solution to new problems.

Kratos is finite element method (FEM) based. Many problems in engineering and applied science are governed by Partial Differential Equations (PDE), easily handled by computer thanks to numerical methods. The FEM is one of the most powerful, flexible and versatile existing methods.

Kratos is object oriented. An integration of disciplines, in the physical as well as in the mathematical sense, suggests the use of the modern object oriented philosophy from the computational point of view. The modular design, hierarchy and abstraction of these approaches fits to the generality, flexibility and reusability required for the current and future challenges in numerical methods.

Kratos is free because is devoted mainly to developers, researchers and students and, therefore, is the most fruitful way to share knowledge and built a robust numerical methods laboratory adapted to their users' needs. Free means you have the freedom to modify and distribute the software.

3.3 GiD - Kratos interaction

GiD pre- and post-processor needs the creation of a problem type to be able to create suitable input data files and to be able to read the Kratos results file. The GiD problem type is the only connection between the pre-processor and the Kratos, such as between the Kratos and the post-processor.

Any GiD Graphical User Interface (GUI) should communicate with Kratos in two ways:

- 1- it must be able to generate the files which are necessary for Kratos to run. These files are usually .mdpa input files that contain geometrical data, and other .py input files that contain parameters/options important for the run.
- 2- It must be able to start the run by calling Python plus a main script.

When the user starts a calculation, GiD runs a .bat file located in the problem type folder. What these files do, essentially, is to set some environment variables to certain values, and when all these variables have been set, call Python to start the calculation.

3.4 Tcl language



Tcl was originally developed by *John Osterhout* in 1988, as a reusable command language for experimental computer aided design (CAD) tools. The source code is compiled into bytecode, which is later interpreted by the Tcl interpreter. The interpreter is implemented as a C library that could be linked into any application. It is very easy to add new functions to the Tcl interpreter, so it is an ideal reusable "macro language" that can be integrated into many applications. However, Tcl is a programming language in its own right, which can be roughly described as a cross-breed between:

- LISP/Scheme (mainly for its tail-recursion capabilities),
- C (control structure keywords, expr syntax),
- Unix shells (but with more powerful structuring).

The name Tcl is derived from "Tool Command Language" and is pronounced "tickle"; it is a string based scripting language, which does not belong to the most popular programming languages in use today. Tcl is a radically simple open-source interpreted programming language that provides common facilities such as variables, procedures, and control structures as well as many useful features that are not found in any other major language. While Tcl is flexible enough to be used in almost any application imaginable, it does excel in a few key areas, including: automated interaction with external programs, embedding as a library into application programs, language design, and general scripting.

The language is commonly used for rapid prototyping, scripted applications, GUIs, and testing. The first major GUI extension that works with Tcl is Tk, a toolkit that aims to rapid GUI development; that is why Tcl is now more commonly called Tcl/Tk.

The language features far-reaching introspection, and the syntax, while simple, is very different from the Fortran/Algol/C++/Java world. Although Tcl is a string based language there are quite a few object-oriented extensions for it like Snit, incr Tcl, and XOTcl to name a few.

The main Tcl features are:

- All operations are commands, including language structures. They are written in prefix notation.
- Everything can be dynamically redefined and overridden.
- All data types can be manipulated as strings, including source code.
- All commands defined by Tcl itself generate error messages on incorrect usage.
- Extensibility, via C, C++, Java, and Tcl.
- Full Unicode support, first released in 1999.

3.5 Python language



Python is a widely used general-purpose, high-level programming language. Its design philosophy emphasises code readability and its syntax allows programmers to express concepts in fewer lines of code than would be possible in languages such as C+ or Java. The language provides constructs intended to enable clear programs on both a small and large scale.

Python supports multiple programming paradigms, including object-oriented, imperative and functional programming or procedural styles. It features a dynamic type system and automatic memory management and has a large and comprehensive standard library.

Python was conceived in the late 1980s and its implementation was starts in December 1989 by Guido van Rossum at CWI (Centrum Wiscunde & Informatica) in the Netherlands as a successor to the ABC language capable of exception handling and interfacing with the Amoeba operating system. Python 2.0 was released on 10 October 2000, and included many major new features including a full garbage collector and support for Unicode. With this release the development process was changed and became more transparent and community-backed. Python 3.0 (also called Python 3000 or py3k), a major, backwards-incompatible release, was released on 3 December 2008 after a long period of testing. Many of its major features have been backported to the backwards-compatible Python 2.6 and 2.7.

Python is a multi-paradigm programming language: object-oriented programming and structured programming are fully supported, and there are a number of language features which support functional programming and aspect-oriented programming. Many other paradigms are supported using extensions, including design by contract and logic programming.

Python uses dynamic typing and a combination of reference counting and a cycle-detecting garbage collector for memory management. An important feature of Python is the dynamic name resolution (late binding), which binds methods and variable names during program execution.

Rather than requiring all desired functionality to be built into the language's core, Python was designed to be highly extensible: a small core language, with a large standard library, is supported by an easily extensible interpreter. Python can also be embedded in existing applications that need a programmable interface.

CHAPTER 4

MODELLING STRATEGY OF X-LAM BUILDINGS AND CONVENTIONS

This Chapter provides a description of the philosophy adopted for modelling Cross-Lam buildings; it is an innovative strategy used in the software. This explanation is the main point to understand the development of the interface and the programming code implemented for the pre-process, which will be presented in the following chapters. Modelling of panels and connections is detailed along the Chapter; finally, conventions about units of measurement and gravity are presented.

4.1 Modelling strategy

The most convenient strategy for modelling X-Lam structures has to be defined. Such strategy must be suitable for automatic generation of numerical models and must have the ability of keeping into account all the possible typologies of connections and assembling of Cross-Lam panels.

The finite element modelling technique, adopted in the developed software, concerns the modelling of panels, connections and curbs. The panels are modelled by shell elements, with orthotropic linear elastic constitutive law. The connections are modelled by punctual spring elements, with isotropic linear elastic constitutive law. The curbs are modelled by beam elements, with isotropic linear elastic constitutive law.

The modelling philosophy consists in the automatic creation of the connections between X-Lam panels. To achieve this goal, an abstract offset is applied between each border shell in which a surface is discretized and the beam elements in which a line is discretized. The offset implies the duplication of the nodes that belong to both beams and shells. It has zero distance to allow an easy management of the nodes, since the

duplicated nodes will have the same coordinates of the original ones. An example of the offset is shown in the introductory Chapter, in Figure 1.1.

The information about the connection properties is set at interface level to the lines, which are then discretized in beams (numeric entities). Therefore, spring elements, with the stiffness values inserted by the user at the interface, are used to join nodes with equal coordinates. The beams are fake elements because they are not necessary for the modelling and analysis of the structure, apart from those set as curbs at the interface. The fake beams must have geometric and structural properties so that their presence is negligible in the analysis. Experimental tests have testified that a proper section could be equal or less than $10^{-4} \cdot 10^{-4} \text{ m}^2$. Hence, fake beams are only used for the automatic process of nodes duplication and springs creation.

4.2 Modelling of X-Lam panels

X-Lam panels are considered as an orthotropic linear elastic material; this particular property is mainly due to two concurrent factors:

- X-Lam panels generally have, in a generic cross section, different values of the total thicknesses t_1 and t_2 in the two directions of the medium plane which defines the panel itself (Figures 4.1 and 4.2); the total thicknesses t_1 and t_2 are defined as the sum of the thicknesses of the layers with the grain oriented in the direction defined by the subscript.
- The values of the elastic modulus of the timber in the direction orthogonal to the grain, as highlighted in Tables 2.4 and 2.5, are one order of magnitude lower than the same module evaluated in parallel to the grain.

Considering the gravity in Z-direction (Section 4.5), the local axes of the orthotropic shell implemented in Kratos are identified in the following way:

- If the panel lies on the global X-Z or Y-Z planes (wall panel):
 - the 3-axis (local z-axis) is identified by the normal of the panel;

- the 1-axis (local x-axis) is identified by the vector product between global Z-axis and 3-axis.
 - the 2-axis (local y-axis) is identified by the vector product between 3-axis and 1-axis.
- If the panel lies on the global X-Y plane (floor panel):
- the 3-axis (local z-axis) corresponds to the global Z-axis;
 - the 1-axis (local x-axis) corresponds to the global X-axis;
 - the 2-axis (local y-axis) corresponds to the global Y-axis.

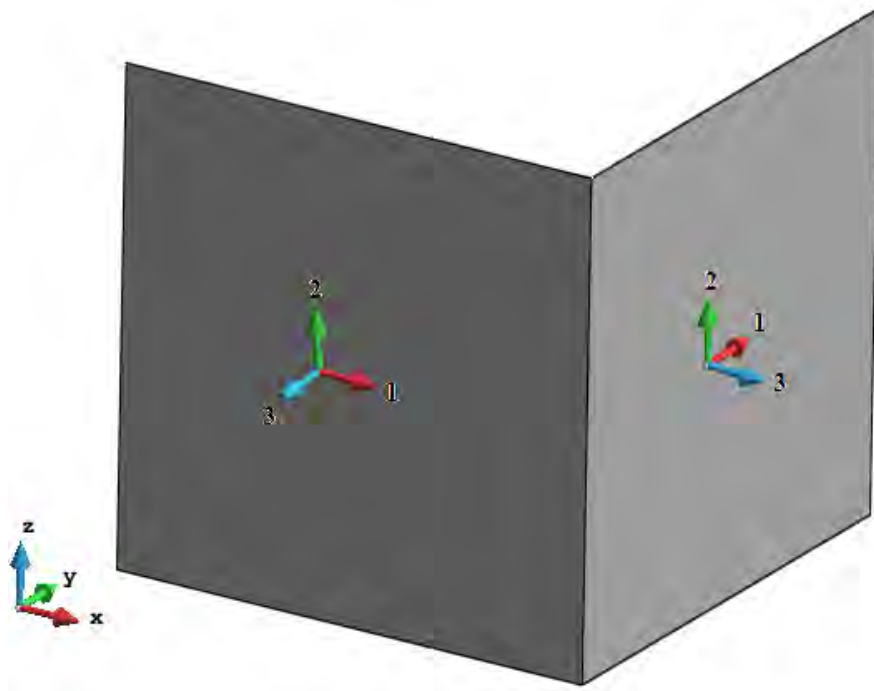


Figure 4.1 Orientation of local axes in wall X-Lam panels

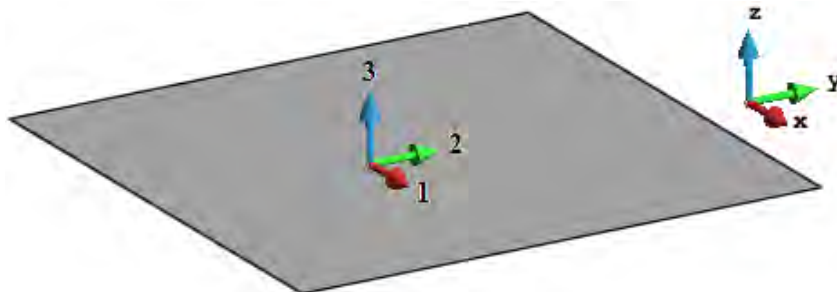


Figure 4.2 Orientation of local axes in a floor X-Lam panel

To fully define the material of an orthotropic shell, three elastic modulus (E_{11} , E_{22} and E_{33}) and three Poisson ratios (ν_{12} , ν_{13} and ν_{23}) should be provided. Three shear moduli (G_{12} , G_{13} and G_{23}) are automatically calculated by means of the previous six parameters.

As already seen in Tables 2.4 and 2.5, X-Lam panels are identified by the elastic moduli E_0 and E_{90} , where subscripts 0 and 90 indicate the direction parallel to the grain and the orthogonal one, respectively.

To relate the elastic moduli of the orthotropic shell with the X-Lam panel ones, the grain orientation of the outer layer of the panel should be taken into account. It is assumed that, if the grain of the outer layer is arranged according to 1-axis (local x-axis), the angle of orientation is equal to zero, otherwise it is ninety degrees. Being the other layers arranged transversely relative to the previous one and identified the orientation of the outer layer, the orientations of the other ones are directly known. Moreover, since the number of layers is always odd, the two sides of the panel can be interchangeably used as reference for the outer layer.

Considering the Kratos convention on the local axes orientation and the convention about the angle, if the angle of grain orientation of the outer layer of the X-Lam panel is zero, modulus E_{11} corresponds to the elastic modulus E_0 , while moduli E_{22} and E_{33} correspond to E_{90} . On the contrary, if the angle of grain orientation of the outer layer of the X-Lam panel is ninety degrees, modulus E_{22} corresponds to the elastic modulus E_0 , while moduli E_{11} and E_{33} correspond to E_{90} .

As it will be shown in Chapter 5 (Section 5.7), the different classes of timber defined by the standard EN 338: 2004, are predefined in the GiD interface; they are all set so that the angle of grain orientation of the outer layer is equal to zero. If it is not zero, the material should not be changed, but it should be set the angle equal to ninety degrees instead of zero; the elastic moduli are directly changed in the calculation of the orthotropic shell. How the angle of grain orientation of the outer layer can be inserted in the interface is explained in detail in Section 5.5.3.

4.3 Connections modelling

The kinematic behaviour of the connections and their stiffness and resistance calculation have already been described in Chapter 2. However, the modelled hold-down stiffness deserves special care because is not the same of the real one, suggested by the code. This comes from the difference in behaviour, under the action of horizontal forces, of a single modelled panel compared with the same in a real situation.

A real panel, subjected to a horizontal force at the top because of a seismic action, neglecting the effect of horizontal translation, tends to rotate around its edge. This happens because the panel is in contact with the soil and the hold-down does not resist to compression. The interaction between the panel and the soil can be considered a contact problem. Alternatively, the problem can be considered non-linear for the material, considering the hold-down as a material resistant to compression and the soil as a material non-reactive to traction. The use of linear elastic constitutive law for the springs in the model leads to the impossibility of simulating the problem in its own non-linearity. Therefore, a modelled panel subjected to the same horizontal force rotates around the mid-point of the bottom side.

This difference in behaviour between real and modelled panel leads to the need of modelling the hold-down stiffness with a different value compared to the real one. For further information about the behaviour of an X-Lam panel subjected to horizontal forces, see the thesis “*Una procedura numerica per il progetto di edifici in X-Lam*” by Massimiliano Zecchetto.

With reference to Figure 4.3, the stiffness value $K_{HD,mod}$ of a modelled hold-down can be obtained by imposing the effects of rotation at the base of the panel, in terms of displacement d_x , to be the same of a real panel subjected to equal overturning moment. With reference to Figure 4.3b, for a modelled panel subjected to a given moment M , follows:

$$d_{y,mod} = \vartheta_{mod} \cdot \frac{B}{2} \quad \rightarrow \quad \vartheta_{mod} = 2 \cdot d_{y,mod} \cdot \frac{1}{B}$$

$$d_{y,mod} = \frac{F_{HD}}{K_{HD,mod}}$$

$$d_{x,mod} = \vartheta_{mod} \cdot H \quad \rightarrow \quad d_{x,mod} = 2 \cdot d_{y,mod} \cdot \frac{H}{B}$$

For a real panel, subjected to the action of the same moment M (Figure 4.3a), follows:

$$d_{y,real} = \vartheta_{real} \cdot B \quad \rightarrow \quad \vartheta_{real} = d_{y,real} \cdot \frac{1}{B}$$

$$d_{y,real} = \frac{F_{HD}}{K_{HD,real}}$$

$$d_{x,real} = \vartheta_{real} \cdot H \quad \rightarrow \quad d_{x,real} = d_{y,real} \cdot \frac{H}{B}$$

To finally get equal components of horizontal displacement in the two cases (modelled and real panel), the value of $K_{HD,mod}$ can be obtained by means of simple algebraic steps:

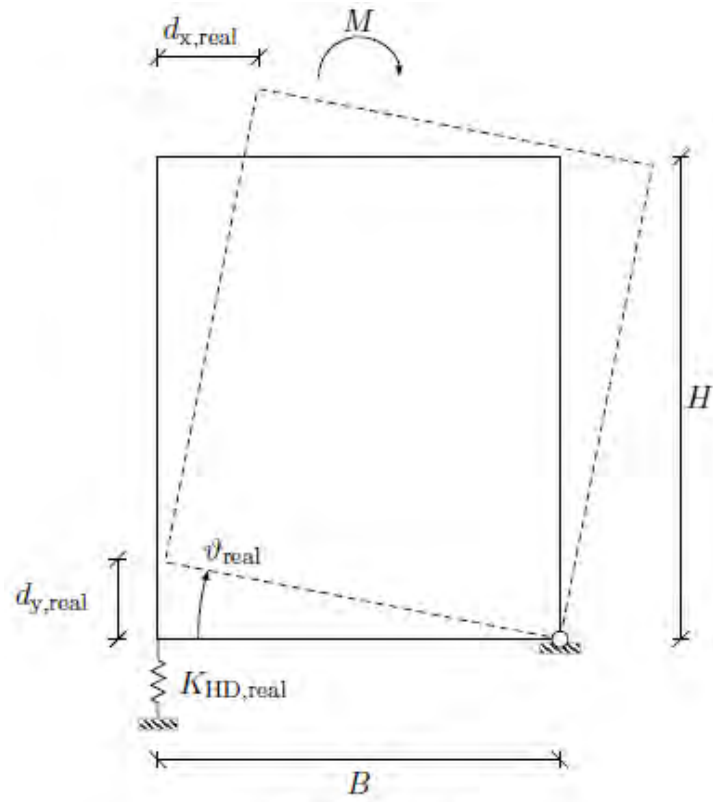
$$d_{x,real} = d_{x,mod}$$

$$d_{y,real} \cdot \frac{H}{B} = 2 \cdot d_{y,mod} \cdot \frac{H}{B}$$

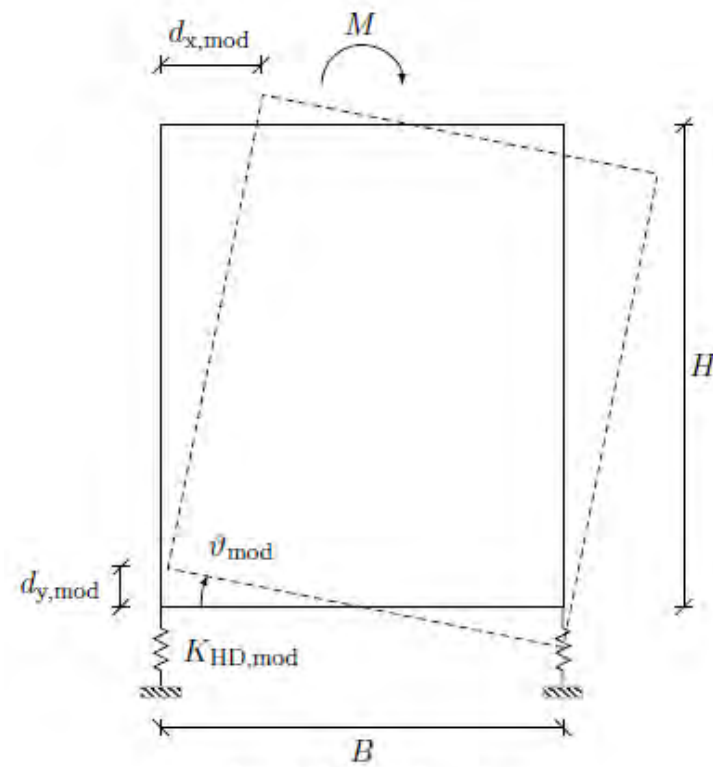
$$\frac{F_{HD}}{K_{HD,real}} = 2 \cdot \frac{F_{HD}}{K_{HD,mod}}$$

$$K_{HD,mod} = 2 \cdot K_{HD,real}$$

Ultimately, the hold-down stiffness used in the model is equal to two times the real one, to properly take into account the difference in behaviour between a real and modelled panel.



(a) Real behaviour



(b) Modelled behaviour

Figure 4.3 Difference in behaviour between a real and a modelled panel

4.4 Units of measurement convention

Relatively to the units of measurement, it is assumed that all parameters are expressed in Newton - meters. This is because Kratos works with these units, and not with those of the international system. All the values inserted by the user should be set with these units of measurement, as well as all the parameters already set refer to this assumption.

4.5 Gravity convention

The gravity convention lies into adopt the Z-axis as vertical one. This is mainly due to two computational reasons:

- the way of identification of the horizontal and vertical beams in the *xlam-driver* application, by Gabriele D’Aronco (see the thesis “*An algorithm for numerical modelling of Cross-Laminated Timber structures*”, Gabriele D’Aronco);
- the convention about the local axes orientation of the orthotropic shells used for modelling panels (Section 4.2).

The gravity convention is important for the correct functioning of the software, as it is till now. Because of the convention, the user must use Z as vertical axis during the geometry creation (Section 5.4). The removal of this restraint is one of the aspects that could be improved in future development of the software.

CHAPTER 5

PRE-PROCESSOR AND INTERFACE TUTORIAL

This Chapter focuses on the pre-processor at interface level, providing a user tutorial. It is clear that the actual creation of a new problem type is not at the interface level, since this is just the result of a programming code. The interface is the only part visible to the user and understandable by all types of users, but everyone should know that the graphical feedback of the interface is only possible by means of a programming code.

The pre-processor at interface level precedes the relative programming code (Chapter 6) because it is simpler to understand firstly the features of the new problem type with a graphical feedback and, then, how these features have been implemented. Moreover, the programming code implemented is probably not suitable to all types of users, whilst the interface is.

5.1 Introduction

GiD pre- and post-processor needs the creation of a problem type to be able to create suitable input data files and to be able to read the Kratos results file. The GiD problem type is the only connection between the pre-processor and the Kratos, such as between the Kratos and the post-processor. A problem type is a set of files configured by a solver developer so that the program can prepare data to be analysed.

Thus, a new problem type has been created with the current goal of analysing an X-Lam structure and with the prospective one of calculating the optimal connections between panels. In order to analyse the features of the new problem type, any part of the menu will be explained in the following sections by means of a simple example, which makes the understanding very easy.

5.2 Example introduction

To understand how the new problem type works, a simple example will be used; all the steps will be explained in detail, to enable the user to reproduce the same example without any uncertainty. The following sections will be a tutorial for learning the basics and advanced features of the new problem type, covering full flow of GiD pre-processing.

The philosophy of this tutorial is to get familiarised with *xlam-kratos* problem type: how to select the problem type, draw the geometry, deal with the IDs, assign the connection properties to the lines, and assign the material and other basic features. Some of these features are both in the pre-processing and the post-processing parts of GiD, although the example that will be shown is from the pre-processing one.

The example will develop a finite element problem in one of its principal phases, the pre-process, and will include the consequent data and parameter description of the problem. This example introduces creation, manipulation and meshing of the geometrical entities used in GiD. To make the comprehension simpler, it will be explained with the help of images, adapted from the GiD interface, which enables to avoid misunderstandings and allows repeating the same example with the only use of the manual, without need of other basic knowledge.

The example presented in this tutorial consists of a simple geometry of 4 panels, not a whole X-Lam structure. Since the goal is not the analysis, but the explanation of the problem type and to allow a better understanding, a trivial geometry will be used. Thus, the main aim is to understand the main features of the problem type, without burdening the description and avoiding misunderstandings. It is clear that the problem type *xlam-kratos* works in the same way when applied to a more complicated problem, like a real X-Lam structure. For this reason, a very simple example, not corresponding to a real structure, is now displayed; conversely, a more complex example will be presented in Chapter 7.

5.3 Problem type selection

GiD is an evolving program, meaning that many different versions are available from the official website; it is always recommended to download the last one, because any updating is an improvement of the previous one.

Once opened the desired version of GiD, the first step is the selection of the problem type by the top command line, as in Figure 5.1.

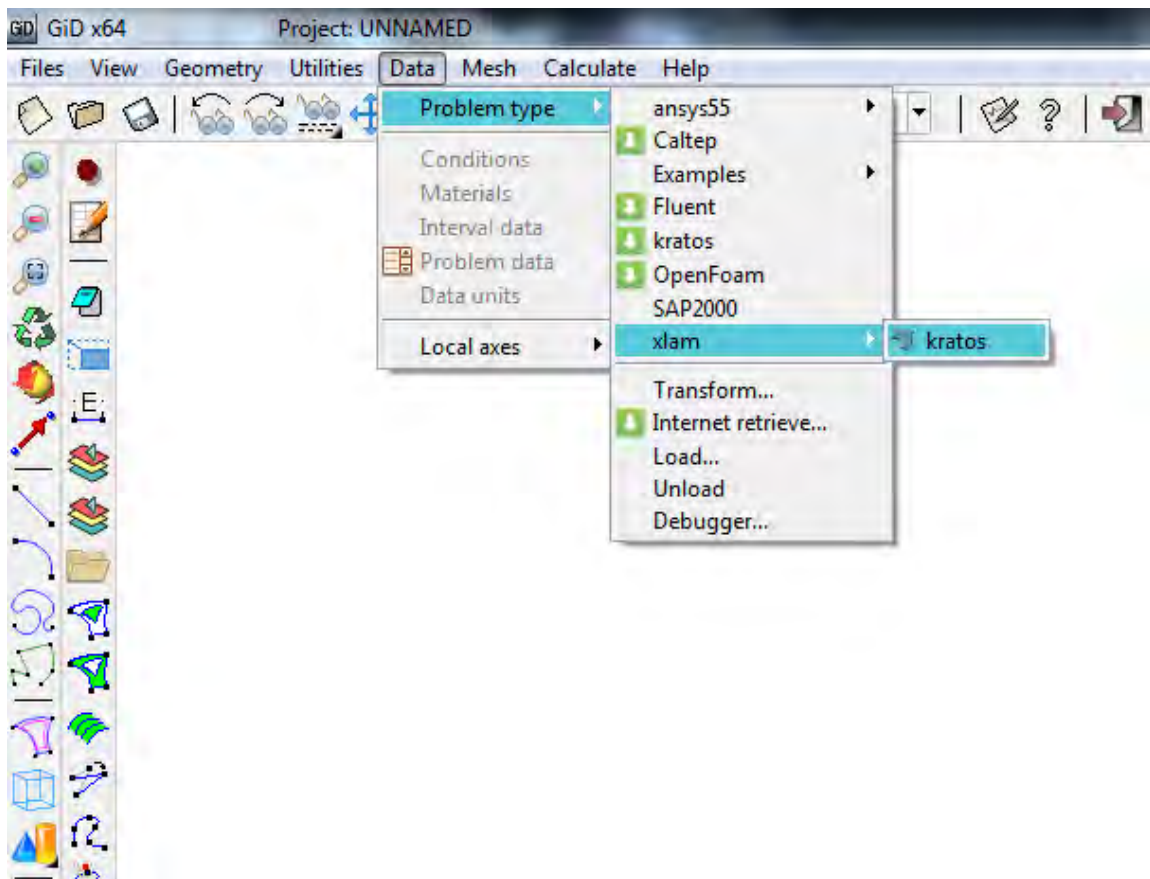


Figure 5.1 Problem type selection

The problem type will appear in the top command line only if it is saved in the folder of the corresponding version of the GiD problem types. The problem type specifically created for the analysis of X-Lam structures is *xlam-kratos* (it uses the solver Kratos); it is saved in the special folder, so that it appears in the list of the problem types as shown in Figure 5.2.

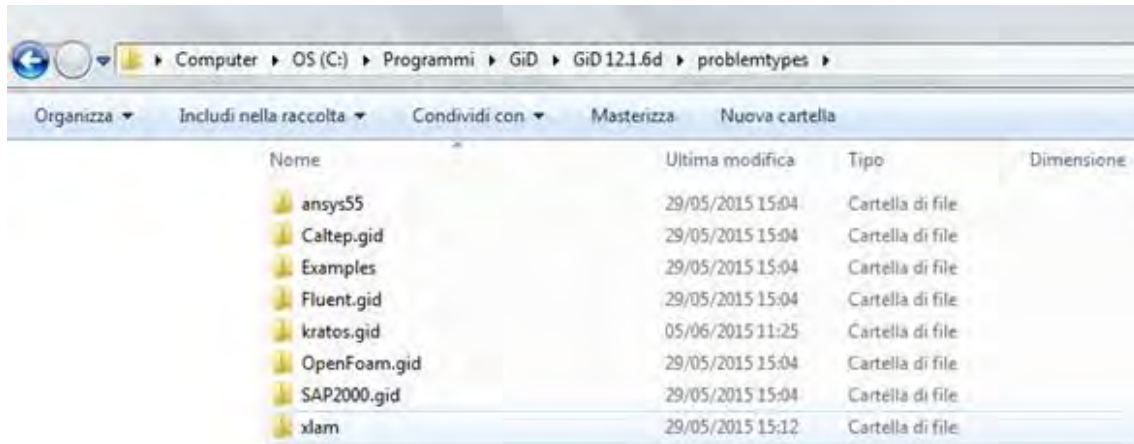


Figure 5.2 GiD problem types folder

Choosing the problem type, another set of menu will appear in the GiD interface: this is the main menu of the selected problem type. In this case of *xlam-kratos* problem type, the new command line will be the one shown in Figure 5.3: the first two columns belong to the GiD menu, while the third column is the *xlam-kratos* menu.

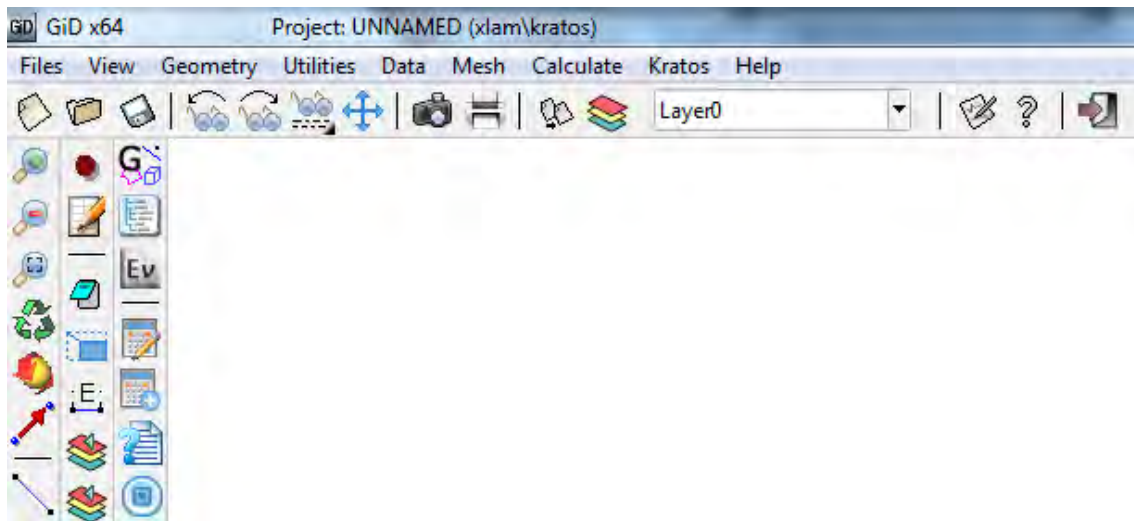


Figure 5.3 “Xlam-kratos” command menu

The choice of the *xlam-kratos* problem type leads the appearance of a window, which remembers to the user to use the z-axis for gravity (Figure 5.4).

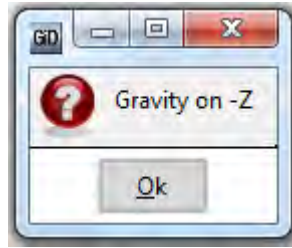


Figure 5.4 Gravity warning

The reasons for recommending the use of the z-axis as vertical one are explained in Section 4.5; the warning window is just to remember it to the user, before he starts drawing.

5.4 Geometry creation

As mentioned in the introduction, the geometry that will be created consists of only 4 panels: two in the plane x-y, one in the plane x-z and another in the plane y-z. In this Section all the steps for its creation will be explained in detail.



Figure 5.5 "Create object" shortcut

The first surface can be created using the command "Create line" and defining the four corner points; after that it would be necessary to use the command "Create NURBS surface", to assign a surface to the four lines created. In this case the surface is created in an easier way, using the command "Create object" on the left command line

of GiD (Figure 5.5), and defining the two corner points with coordinates (0, 0) and (1, 1). Being the units of measurement in meters (reference is made to Section 4.4), this means to create a surface of 1 m². Figure 5.6 shows the first surface created in the plane x-y.

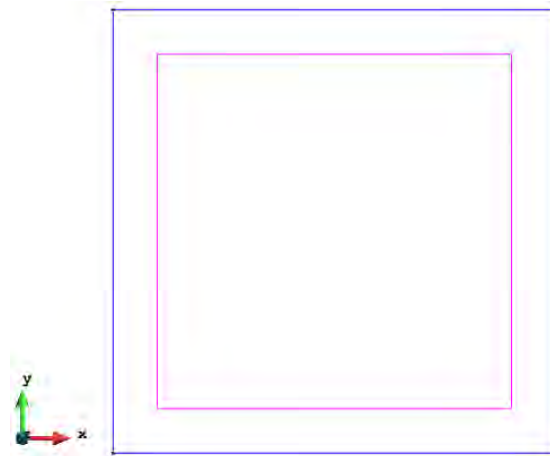


Figure 5.6 *First surface*

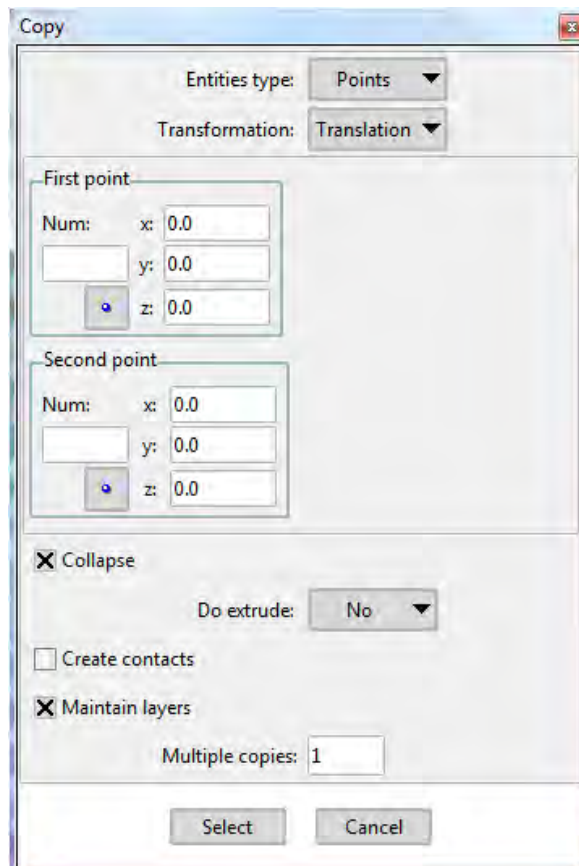


Figure 5.7 *“Copy” menu*

The other surfaces can be created using the commands “*Create line*” and “*Create NURBS surface*” as explained before, or following the same procedure used before, by means of the command “*Create object*”. In this case the other surfaces are created copying the first one in different directions: it is an easier and fastest way and it also allows the use of other GiD commands. Using the top command line and selecting “*Utilities - Copy*”, the menu of Figure 5.7 will appear.

In order to create other two surfaces from lines already created, lines should be copied and surfaces extruded; it means to choose “*Lines*” as “*Entities type*” and to select “*Surfaces*” in the line “*Do extrude*” (reference is made to Figure 5.7). The copy menu modifies as shown in Figure 5.8.

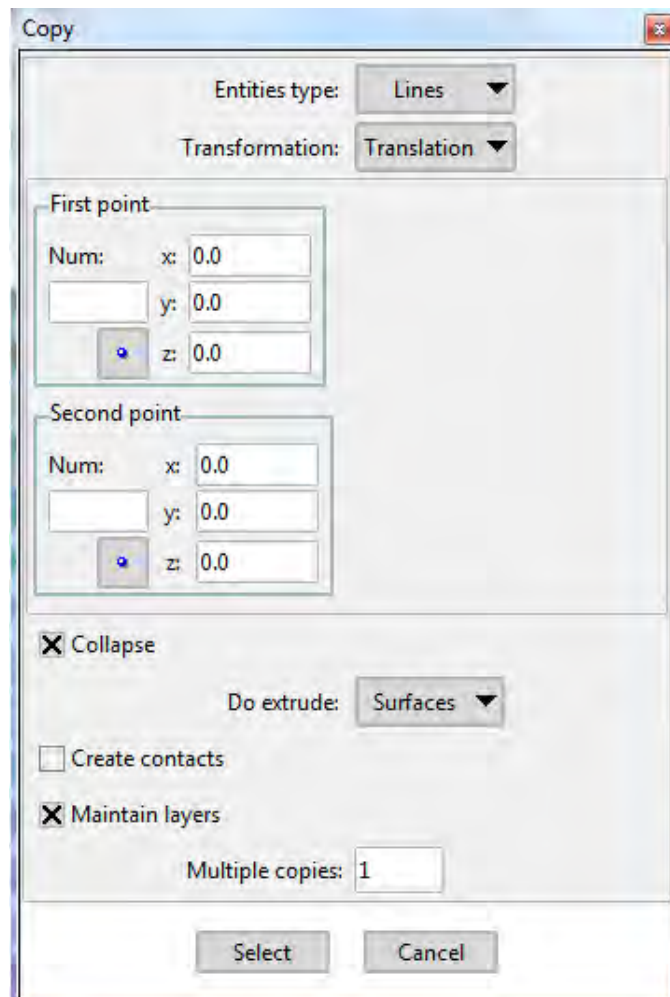


Figure 5.8 “*Copy*” menu for copying surfaces

Thus, another surface can be created, writing 1.0 as z-coordinate of the second point and selecting the top line of the previous surface created; it returns the geometry shown in Figure 5.9.

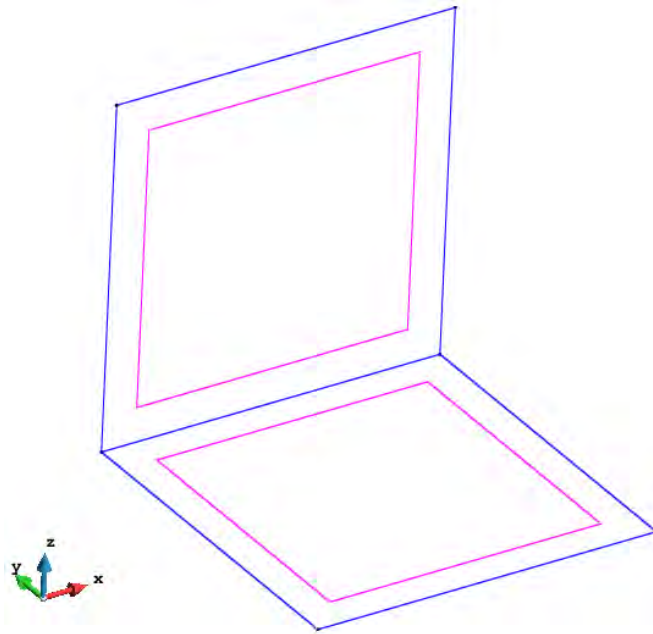


Figure 5.9 *First and second surfaces*

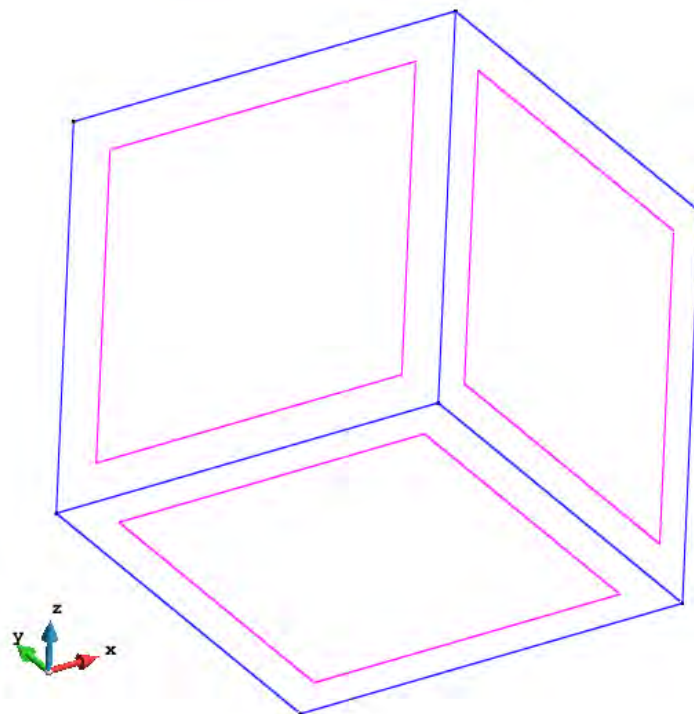


Figure 5.10 *First, second and third surfaces*

In a similar way, writing 1.0 as z-coordinate of the second point and selecting the right line of the first surface created, another surface can be created; the geometry results as in Figure 5.10.

In order to create another surface copying the first surface in z-direction, “*Surfaces*” should be selected as “*Entities type*” and “*No*” on the line “*Do extrude*” in the copy menu (Figure 5.7). Hence, selecting the first surface created, the final geometry results as in Figure 5.11.

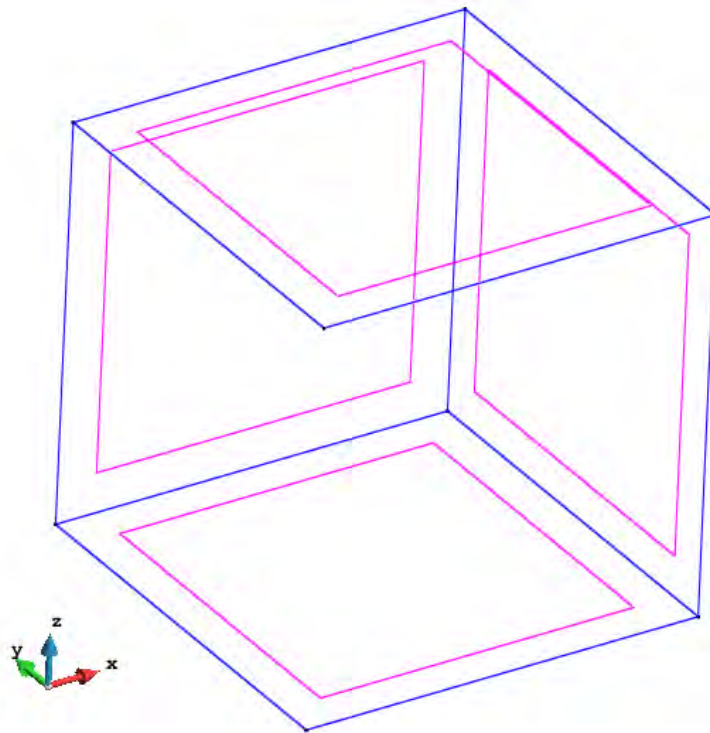


Figure 5.11 End geometry

Even if the goal is not the calculation of the structure, the surfaces have been created considering the gravity on z-axis, as if it was a real example, meaning that the first and last surfaces created will be slabs and the two others walls: the slabs are in the plane x-y and the walls are in the planes x-z and y-z.

In order to understand the following steps, the GiD ID (identification number) labels of each node, line and surface can be visualised, clicking the right mouse button and selecting “*Label - All*”. The IDs of the geometry are shown in Figure 5.12.

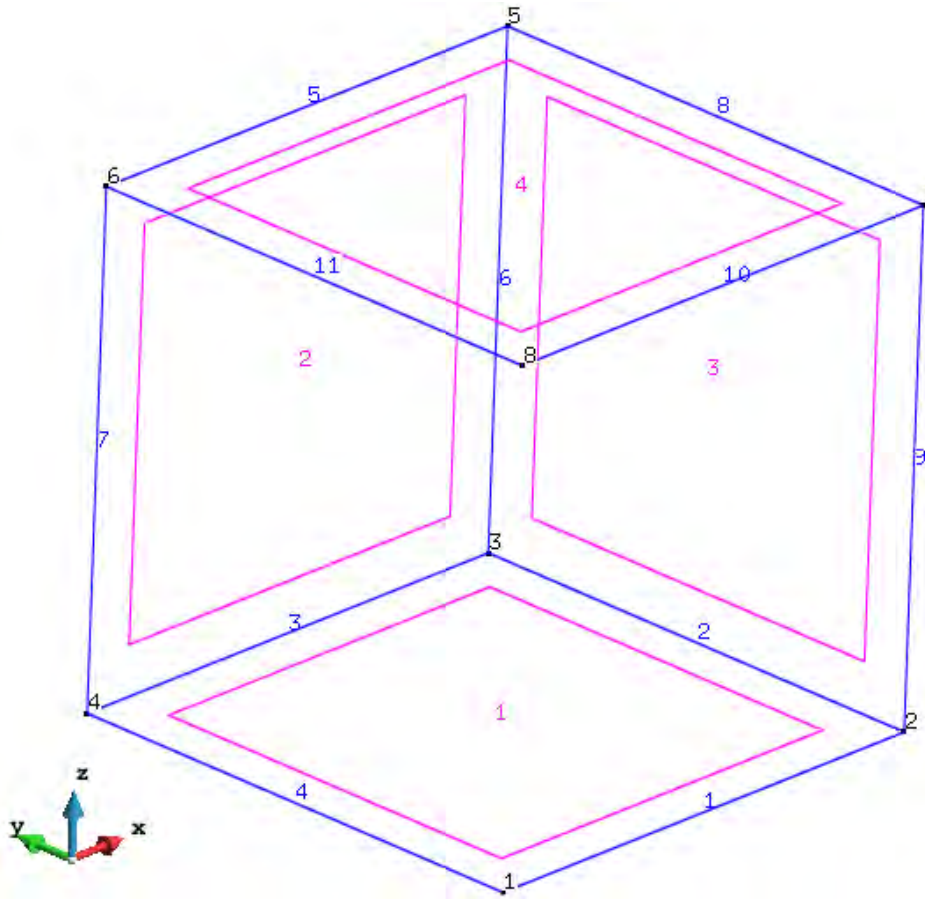


Figure 5.12 *Geometry IDs*

The identification numbers of the nodes are shown in black, of the lines in blue and of the surfaces in pink. These are the original GiD IDs, which are assigned by default by the processor when a geometry is drawn. It is noticeable that the IDs are defined according with the order of the drawing, meaning that, if the surfaces are created in a different order (for example the right line is copied in z-direction before the top one), the identification numbers will be different. As a matter of fact, the end geometry could have also been created copying the lines at the same time, without selecting them one at a time, but in this way the order of creation of the surfaces would have been random, meaning that not necessarily the IDs would be the same. For this reason it is recommended to follow the same procedure and order used in this example, in order to better understand the following steps, since the IDs play an important role.

5.5 Model properties definition

The second icon of the *xlam-kratos* menu enables the definition of the model properties; clicking on it, the window shown in Figure 5.13 will appear.

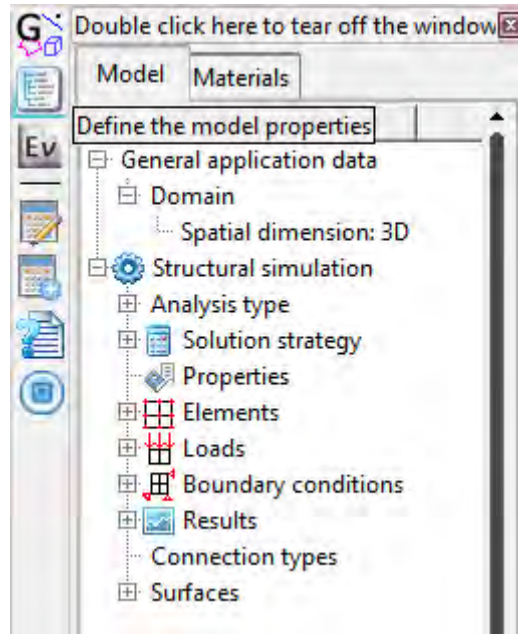


Figure 5.13 “Model properties” menu

This menu allows to define different properties, each one of them will be discussed in detail in the following sections. They will not be explained in the order in which they appear, but in the order that in general the user will follow for the solution of a problem.

5.5.1 Connection properties creation

The connection properties can be defined in the lower section of the model properties menu. The first step is the definition of the “*Connection mode*”, which can be “*Custom*” or “*Standard*”; as default, it is set a custom mode, but it can be changed simply clicking on “*Connection mode*”, and selecting the other one. The difference between the two modes lays in the way of defining the connections. The standard menu allows to select a type of connection already default by a database; conversely, the custom menu allows to assign some parameters, through which the stiffness are

calculated, or to directly set the stiffness. In general, the standard mode is recommended if all connections have already been calculated in detail and the analysis is used only for verification. Differently, the custom one is recommended in case of first attempt connections or to assign possibly even random stiffness, when the analysis is used to determine the optimal connections. At this stage it would be complex to decide which mode to use; thus, before selecting one for the example, both the modes will be explained.

The custom menu is shown in Figure 5.14. Inside the “*Custom Connections*”, there is a sub-menu of “*Custom Property*”, which in turn contains a property already defined, called “*Continuity connection*”.

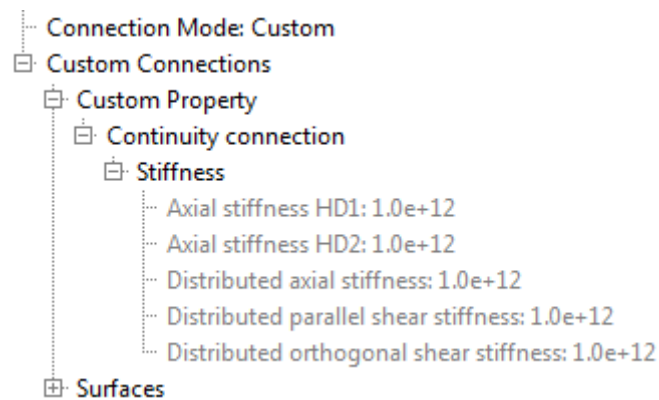


Figure 5.14 *Custom mode menu*

The connection property “*Continuity connection*” has all the stiffness set to infinite, or, better, to a very high value, equal to 10^{12} N/m. The choice of the value is arbitrary, appropriately larger than the ordinary connections stiffness. In future development of the software, the stiffness values of the continuity constraints will be defined automatically, properly higher than the stiffer panel of the structure. This type of property can be used, for example, when the user wants to guarantee the continuity between two panels, meaning that actually two surfaces are only one panel, so there is no connection between them. Thus, because it is not obvious that the user draws a surface for each panel, he can draw the surfaces how he prefers; the calculation works well in every case, as shown in the thesis “*An algorithm for numerical modelling of Cross-Laminated Timber structures*” by Gabriele D’Aronco. The parameters of the

connection property “*Continuity connection*” cannot be changed by the user, he should define another connection property in order to assign other stiffness.

Clicking twice on “*Custom Property*”, a new connection property with custom mode can be created; the menu of Figure 5.15 will appear.

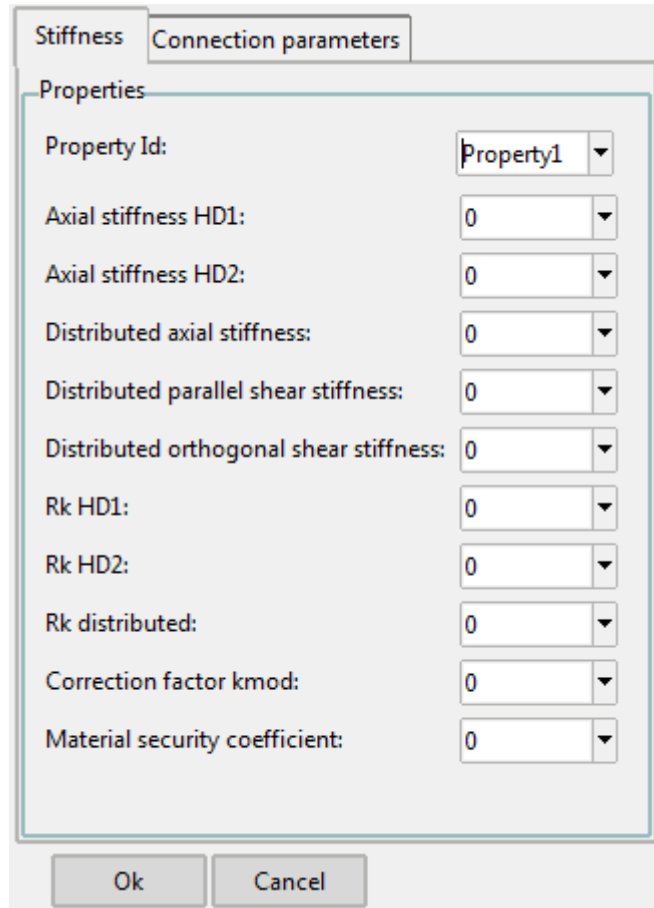


Figure 5.15 “*Stiffness*” menu

The custom property contains in turn two ways to define a connection property: the first one is “*Stiffness*” (Figure 5.15) and the second one is “*Connection parameters*” (Figure 5.16).

The menu “*Stiffness*” requires a new property name and some parameters, which are five stiffness, three load-carrying capacities and two coefficients, which should be introduced by the user. While the stiffness are used for the analysis, the load-carrying capacities and the coefficients come into play only in the verification phase, but they

should be already set at this interface level, meaning in the pre-process, for being available for the post-process. The values that should be set in the menu “*Stiffness*” are:

- Axial stiffness HD1: this is the axial stiffness of the first hold-down, in N/m. The hold-down with less coordinate on the x or y axis is considered as first hold-down, in order to distinguish it from the other one of the panel.
- Axial stiffness HD2: this is the axial stiffness of the second hold-down, in N/m. As second hold-down, it is considered the hold-down with greater coordinate on the x or y axis, in order to distinguish it from the first one. The two hold-down are distinguished to leave the possibility of disposing two different types of hold-down to one and the other side of the panel, or to put only one of them in case of two surfaces which are actually a unique panel (see Section 7.3.3).
- Distributed axial stiffness: this is the axial stiffness of the brackets or distributed nailing, in N/m. This stiffness is in general considered zero because the shear connections do not work in tension (see Section 2.7.1); anyway it is set to enable the user to use a different one, if he wants to assign an axial stiffness to the shear connections or to simulate a distributed axial stiffness (see Section 7.3.3).
- Distributed parallel shear stiffness: this is the shear stiffness in the plane of the panel, in N/m, meaning the brackets' or distributed nailing's shear stiffness.
- Distributed orthogonal shear stiffness: this is the shear stiffness out of the plane of the panel, in N/m.
- R_k HD1: this is the characteristic load-carrying capacity of the first hold-down, in N, which can be calculated as explained in Section 2.7.3.
- R_k HD2: this is the characteristic load-carrying capacity of the second hold-down, in N, which can be calculated as explained in Section 2.7.3.
- R_k distributed: this is the characteristic load-carrying capacity of the brackets or distributed nailing, in N, which can be calculated as explained in Section 2.7.3.
- Correction factor k_{mod} : this is the correction factor taking into account the effects of load duration and moisture content on the timber strength, which is prescribed by the code UNI EN 1995-1-1: 2009 (see Table 2.2).
- Material security coefficient: this is the partial factor for a material property γ_M , which is prescribed by the code UNI EN 1995-1-1: 2009 (see Table 2.3).

Each of the parameters explained presents, on the interface, an abbreviated name that may be not perfectly clear, for a matter of space at the interface. Anyway, placing the mouse over the box corresponding to each of the parameters, pop-ups that explain their meaning will appear.

The stiffness introduced in the processor for the analysis are not only this one, there are actually other three rotational stiffness, hidden to the user, which are set as default to a small value equal to 100 N/m (this small value is to mean zero). Similarly to the “*Continuity connection*” values, the choice of the value is arbitrary, negligible compared to the ordinary connections stiffness. These three stiffness are not defined by the user because they are in general a small value that the user does not calculate every time: they are only necessary to define all the stiffness of the springs that simulate the connections in the *xlam-driver* application (see the thesis “*An algorithm for numerical modelling of Cross-Laminated Timber structures*” by Gabriele D’Aronco).

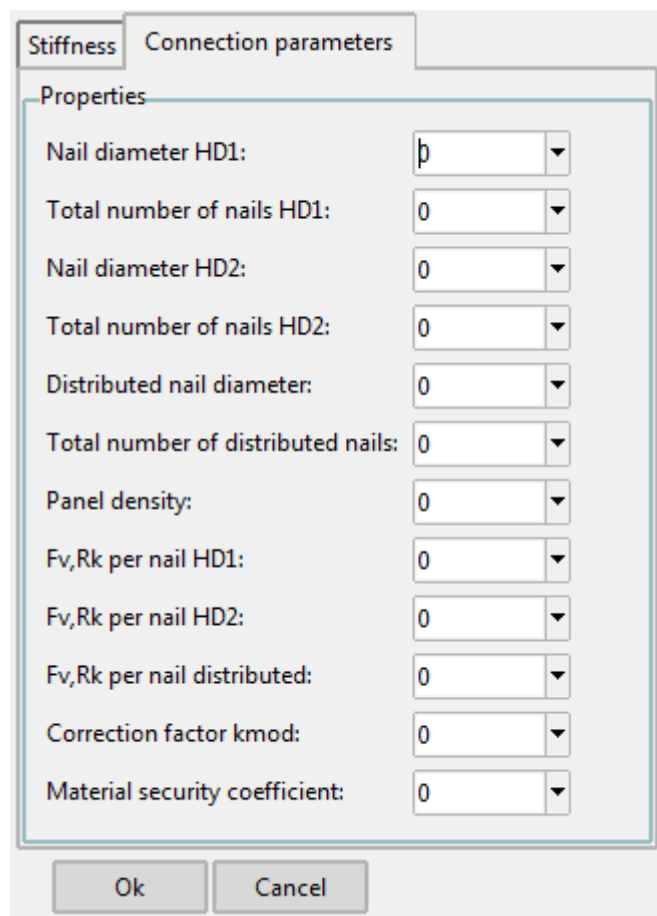


Figure 5.16 “Connection parameters” menu

The menu “*Connection parameters*” (Figure 5.16) requires a new property name and some parameters, through which the stiffness and the design load-carrying capacities are calculated by the processor (as explained in Sections 2.7.2 and 2.7.3). Also in this case the initial parameters are used for the analysis, while the load-carrying capacities and the coefficients come into play only in the verification phase, but the relative parameters should be already set at this interface level, meaning in the pre-process, for being available for the post-process. The values that should be set in the menu “*Connection parameters*” are:

- Nail diameter HD1: this is the nail diameter of the first hold-down, in m (the hold-down with less coordinate on the x or y axis).
- Total number of nails HD1: this is the total number of nails of the first hold-down, meaning the number of nails if only one hold-down is disposed, the number of nails multiplied by the number of hold-down if more than one hold-down are disposed.
- Nail diameter HD2: this is the nail diameter of the second hold-down, in m (the hold-down with greater coordinate on the x or y axis). As for the “*Stiffness*” menu, the two hold-down are distinguished to leave the possibility of disposing two different types of hold-down to one and the other side of the panel, or to put only one of them in case of two surfaces which are actually a unique panel (see Section 7.3.3).
- Total number of nails HD2: this is the total number of nails of the second hold-down, meaning the number of nails if only one hold-down is disposed, the number of nails multiplied by the number of hold-down if more than one hold-down are disposed.
- Distributed nail diameter: this is the nail diameter of the brackets or distributed nailing, in m.
- Total number of distributed nails: this is the total number of nails of the brackets or distributed nailing, meaning the number of nails if only one bracket or a distributed nailing is disposed, the number of nails multiplied by the number of brackets if more than one bracket are disposed.

- Panel density: this is the density of the panels that the connection should joint, in N/m^3 . This density will be also introduced in the definition of the material (see Section 5.7), but it should be set here to enable the calculation of the connection stiffness by the processor.
- $F_{V,Rk}$ per nail HD1: this is the characteristic load-carrying capacity per nail of the first hold-down, in N, which can be calculated as explained in Section 2.7.3.
- $F_{V,Rk}$ per nail HD2: this is the characteristic load-carrying capacity per nail of the second hold-down, in N, which can be calculated as explained in Section 2.7.3.
- $F_{V,Rk}$ per nail distributed: this is the characteristic load-carrying capacity per nail of the brackets or distributed nailing, in N, which can be calculated as explained in Section 2.7.3.
- Correction factor k_{mod} : this is the correction factor taking into account the effects of load duration and moisture content on the timber strength, which is prescribed by the code UNI EN 1995-1-1: 2009 (see Table 2.2).
- Material security coefficient: this is the partial factor for a material property γ_M , which is prescribed by the code UNI EN 1995-1-1: 2009 (see Table 2.3).

Each of the parameters explained presents, on the interface, an abbreviated name that may be not perfectly clear, for reasons of space at the interface. Anyway, placing the mouse over the box corresponding to each of the parameters, pop-ups that explain their meaning will appear.

Also in this case, there are other three rotational stiffness, hidden to the user, which are set as default to a small value equal to 100 N/m (this small value is to mean zero). The choice of the value is arbitrary, negligible compared to the ordinary connections stiffness.

The standard menu is shown in Figure 5.17. Inside the “*Standard Connections*”, there is a sub-menu of “*Standard Property*”, which, as for the custom menu, in turn contains a property already defined, called “*Continuity connection*”.

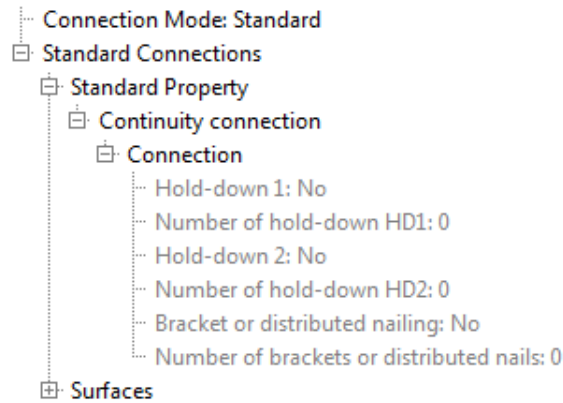


Figure 5.17 Standard mode menu

The connection property “*Continuity connection*” has all the parameters set to zero: it can be used, for example, when the user wants to guarantee the continuity between two panels, meaning that actually two surfaces are only one panel, so there is no connection between them. In case of selection of this connection property, the stiffness are not calculated as for the other standard connections, but they are all set to infinite to guarantee continuity, as for the similar already defined custom property. The parameters of the connection property “*Continuity connection*” cannot be changed by the user, he should define another connection property in order to assign other parameters.

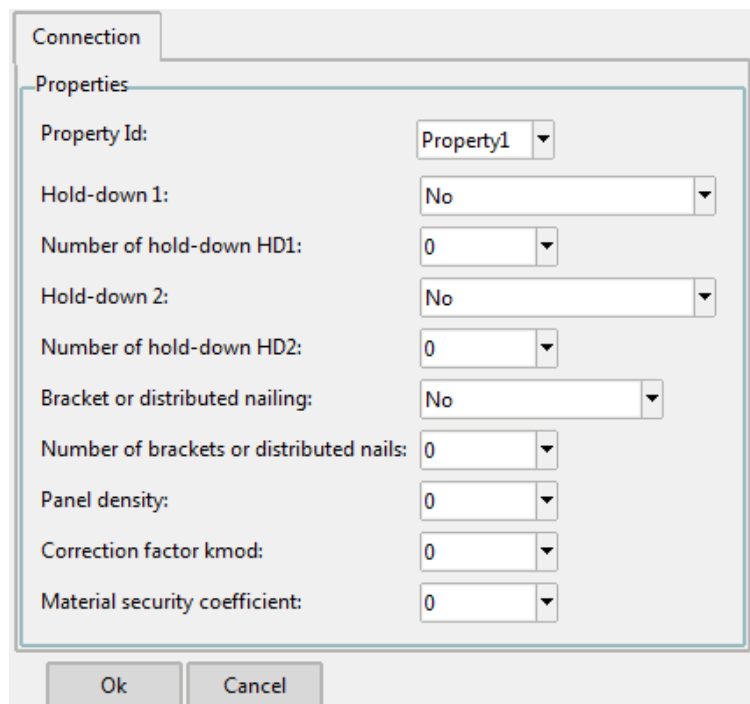


Figure 5.18 “Connection” menu

Clicking twice on “*Standard Property*”, a new connection property with standard mode can be created; the menu of Figure 5.18 will appear.

The standard property contains only one way to define a connection property, “*Connection*”, which requires a new property name and some parameters, which should be introduced by the user. In this case some parameters are only used for the analysis, while other ones are both used for the analysis and the verification. The values that should be set in the menu “*Connection*” are:

- Hold-down 1: this is the type of the first hold-down, which can be selected by an already default database. The database has been created following the Rothoblaas catalogue and distinguishing between total and partial nailing for any type of hold-down; it is shown in Figure 5.19 and it is the same for the first and second hold-down. The hold-down with less coordinate on the x or y axis is considered as first hold-down, in order to distinguish it from the other one of the panel.

```
WHT340 tot nail d4*40
WHT340 tot nail d4*60
WHT340 part nail d4*40
WHT340 part nail d4*60
WHT440 tot nail d4*40
WHT440 tot nail d4*60
WHT440 part nail d4*40
WHT440 part nail d4*60
WHT540 tot nail d4*40
WHT540 tot nail d4*60
WHT540 part nail d4*40
WHT540 part nail d4*60
WHT620 tot nail d4*40
WHT620 tot nail d4*60
WHT620 part nail d4*40
WHT620 part nail d4*60
```

Figure 5.19 *Hold-down database*

- Number of hold-down HD1: this is the number of hold-down for the first hold-down.
- Hold-down 2: this is the type of the second hold-down, which can be selected by an already default database, equal to the one of the first hold-down. The hold-down with greater coordinate on the x or y axis is considered as second hold-

down, in order to distinguish it from the first one. As for the custom menu, the two hold-down are distinguished to leave the possibility of disposing two different types of hold-down to one and the other side of the panel, or to put only one of them in case of two surfaces which are actually a unique panel (see Section 7.3.3).

- Number of hold-down HD2: this is the number of hold-down for the second hold-down.
- Bracket or distributed nailing: this is the type of bracket or distributed nailing, which can be selected by an already default database. The database has been created following the Rothoblaas catalogue and distinguishing between total and partial nailing and concrete-timber or timber-timber for any type of bracket and identifying different lengths of nails for the distributed nailing; it is shown in Figure 5.20.

```
WBR100 tot nail d4*60 conc-tim
WBR100 part nail d4*60 conc-tir
WBO100 tot nail d4*60 conc-tirr
WBO100 part nail d4*60 conc-ti
WBR100 tot nail d4*60 timb-tim
WBR100 part nail d4*60 timb-tir
WBO100 tot nail d4*60 timb-tirr
WBO100 part nail d4*60 timb-tir
diameter 4 length 30
diameter 4 length 35
diameter 4 length 40
diameter 4 length 45
diameter 4 length 50
diameter 4 length 60
diameter 4 length 70
diameter 4 length 80
```

Figure 5.20 *Brackets and distributed nailings database*

- Number of brackets or distributed nails: this is the number of brackets or distributed nails of the shear connection.
- Panel density: this is the density of the panels that the connection should joint, in N/m^3 . This density will be also introduced in the definition of the material (see Section 5.7), but it should be set here to enable the calculation of the connection stiffness by the processor.

- Correction factor k_{mod} : this is the correction factor taking into account the effects of load duration and moisture content on the timber strength, which is prescribed by the code UNI EN 1995-1-1: 2009 (see Table 2.2).
- Material security coefficient: this is the partial factor for a material property γ_M , which is prescribed by the code UNI EN 1995-1-1: 2009 (see Table 2.3).

Each of the parameters explained presents, on the interface, an abbreviated name that may be not perfectly clear, for a matter of space at the interface. Anyway, placing the mouse over the box corresponding to each of the parameters, pop-ups that explain their meaning will appear.

Explained all kinds of menus for the definition of a connection property, it can be now decided which mode will be used, if the “*Custom*” or the “*Standard*” one. For the example, it will be selected the “*Custom*” menu, in order to explain another peculiarity of this menu, considering that it contains two different types of connection property.

The image shows a software dialog box titled 'Stiffness' with a sub-tab 'Connection parameters'. The dialog is used to define properties for a connection. It contains several input fields, each with a dropdown arrow, and two buttons at the bottom: 'Ok' and 'Cancel'.

Parameter	Value
Property Id:	Property1
Axial stiffness HD1:	1.0e+7
Axial stiffness HD2:	1.0e+7
Distributed axial stiffness:	100
Distributed parallel shear stiffness:	1.0e+7
Distributed orthogonal shear stiffness:	100
Rk HD1:	1.0e+5
Rk HD2:	1.0e+5
Rk distributed:	7.0e+4
Correction factor kmod:	1.1
Material security coefficient:	1.3

Figure 5.21 “Property1” parameters

We define two different properties, one belonging to the menu “*Stiffness*” and the other one belonging to the menu “*Connection parameters*”. Both of them will be defined with random parameters because the goal of this simple example is not the analysis, the values have the only purpose of illustrating the assignation. The two properties are named “*Property1*” and “*Property2*” and their parameters are shown in Figures 5.21 and 5.22.

Parameter	Value
Nail diameter HD1:	0.004
Total number of nails HD1:	30
Nail diameter HD2:	0.004
Total number of nails HD2:	30
Distributed nail diameter:	0.004
Total number of distributed nails:	6
Panel density:	3433.5
Fv,Rk per nail HD1:	2000
Fv,Rk per nail HD2:	2000
Fv,Rk per nail distributed:	2000
Correction factor kmod:	1.1
Material security coefficient:	1.3

Figure 5.22 “*Property2*” parameters

Once defined the two properties that will be used in addition to the “*Continuity connection*”, the custom menu appears as in Figure 5.23.

For every new property defined, below the name is the mode to which the connection property belongs: it can be “*Stiffness*” or “*Parameters*”. As default the mode is set on “*Stiffness*”, so the first property does not need any modification of the mode, but for the second property the mode should be changed, clicking on “*Mode*” below its

name, and selecting “*Parameters*”. The selection of the mode is very important because, if it is not selected correctly, the parameters used for the analysis would not be the desired ones, but always the ones of the mode “*Stiffness*” since this is set as default.

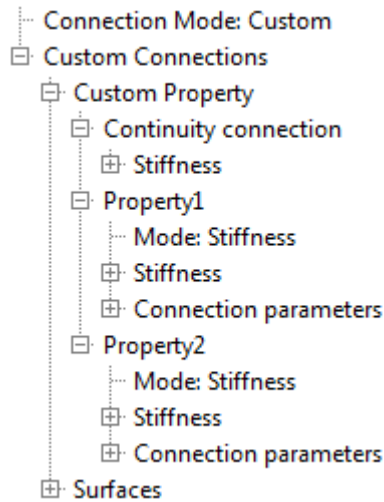


Figure 5.23 Custom menu with all properties

The final custom menu, with all the properties and their modes selected, appears as in Figure 5.24.

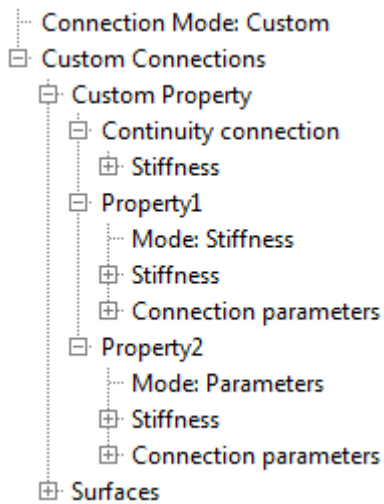


Figure 5.24 Custom menu with all properties and their mode

5.5.2 Connection properties assignation

The end section of the connection properties menu consists on the menu “*Surfaces*”, on which we can focus to then understand how to assign the connection properties. The menu “*Surfaces*” shows a list of all the surfaces of the geometry and, for any surface, the belonging lines; the lines that belong to more than one surface are shown more times, as belonging to the different surfaces they appertain.

In this example, for the geometry created before, the menu “*Surfaces*” is shown in Figure 5.25; Figure 5.26 shows the geometry with its ID labels, in order to quickly identify the surfaces and lines shown in the “*Surfaces*” menu.

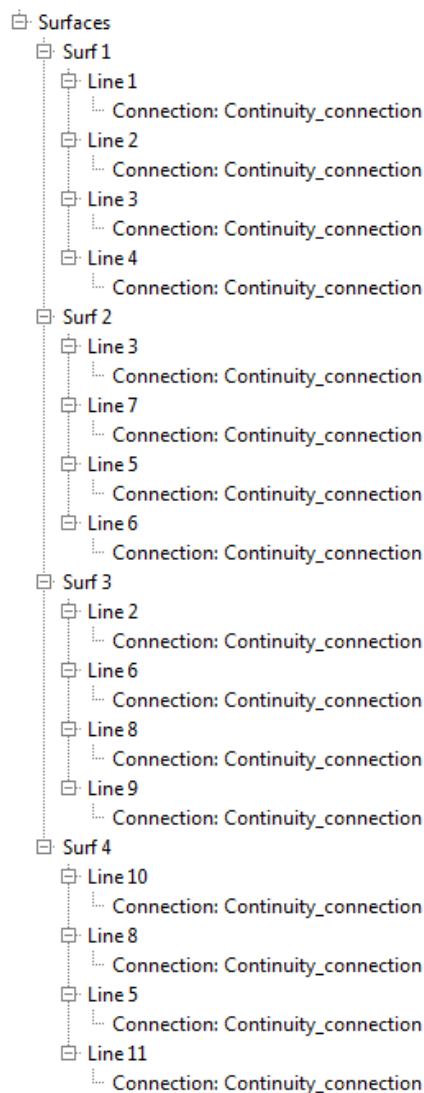


Figure 5.25 “*Surfaces*” menu

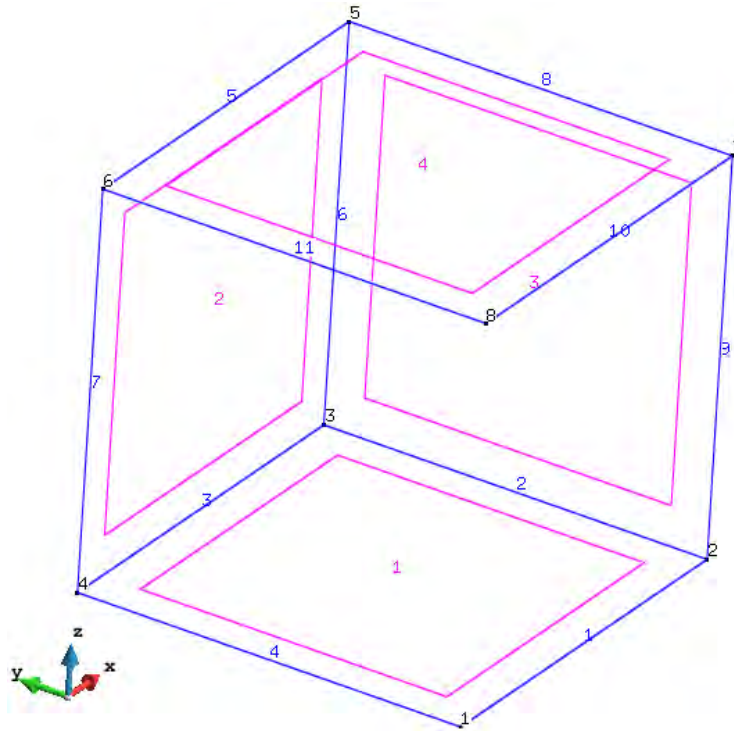


Figure 5.26 Geometry created before

Checking the geometry and the “Surfaces” menu, it is clear that the four surfaces and all the lines belonging to them are shown in the tree.

Pretending that one of the surfaces was drawn by mistake, and it needs to be removed, it can be removed selecting the command “Delete” (Figure 5.27) at the bottom section of the GiD menu. In particular, the command that allows removing at the same time surfaces, lines and points could be useful: for instance, when removing surface 1, lines 1 and 4 and point 1, which do not belong to any other surface, will be removed as well.

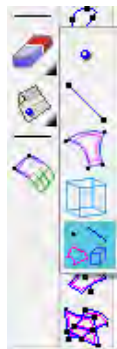


Figure 5.27 “Delete” shortcut

Thus, selecting surface, lines and point previously mentioned, they can be all removed at the same time. Following removal of these entities, the list of surfaces and lines previously shown changes: surface 1 will not appear, as well as lines 1 and 4. Obviously, lines 2 and 3 have not been removed because they also belong respectively to surfaces 3 and 2: GiD has a hierarchy of entities, for which lines cannot be removed if all surfaces containing them have not been removed first. Likewise, the same happens for lines and points and for surfaces and volumes. The new list of surfaces and lines is shown in Figure 5.28, while the new geometry is shown in Figure 5.29.

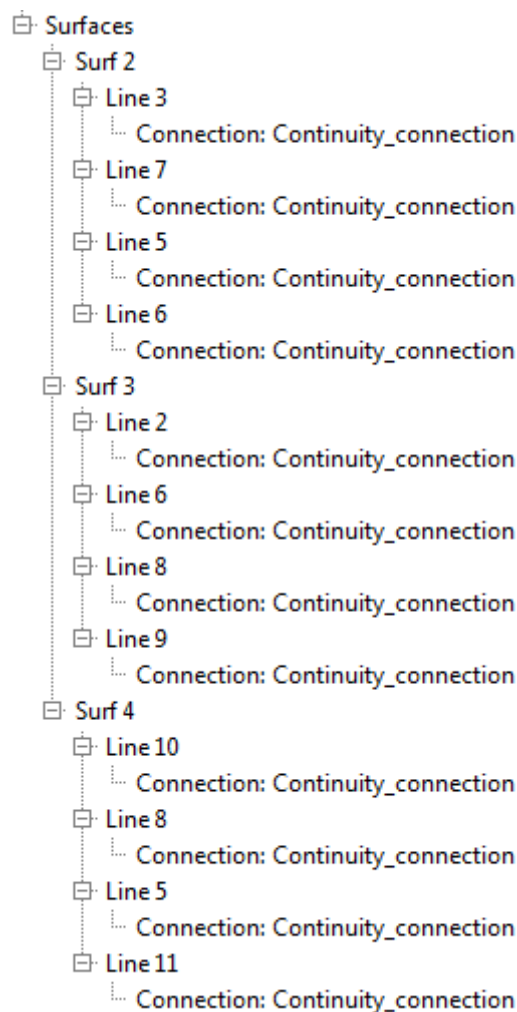


Figure 5.28 End “Surfaces” menu

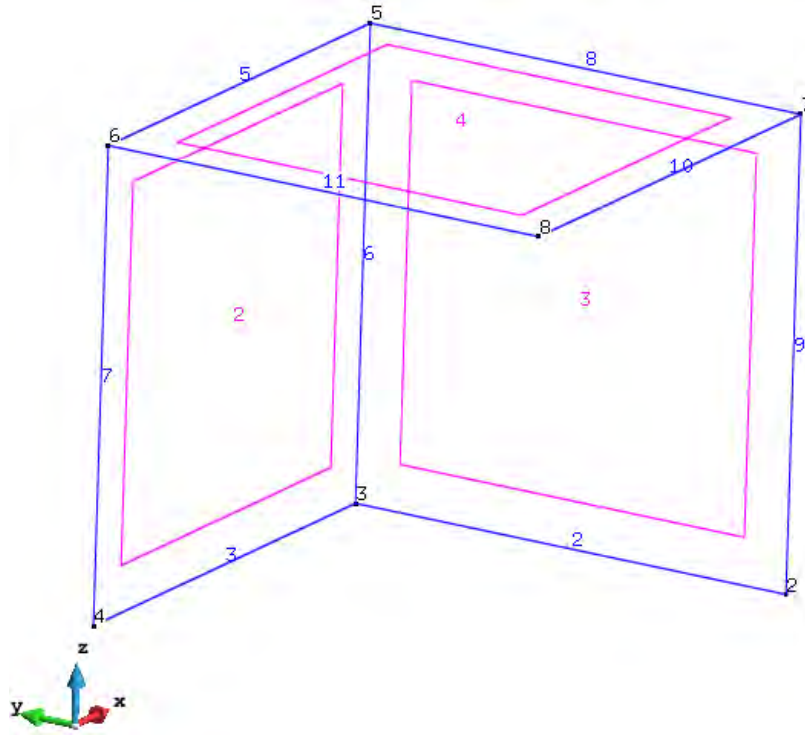


Figure 5.29 End geometry

As expected, surface 1 has been removed from the tree, as well as the lines belonging to it; lines 1 and 4 do not appear in the tree because they only belonged to surface 1, while lines 2 and 3 persist in the list as belonging respectively to surfaces 3 and 2. All the IDs mentioned so far are the GiD IDs, dependent on the order of drawing of the geometry, as already explained; currently we refer to them every time we mention the IDs.

The “*Surfaces*” menu just described enables to assign the previously created connection properties to the lines; they are assigned to the lines because it is simpler for the user to select a line and to assign it the properties of the connection he wants on this side of the panel. For the pre-process, the connection remains a property of the line, because we only care about geometrical entities; then, in the *xlam-driver* application, the stiffness will be assigned to the springs (see Section 4.1). Thereafter, these springs will assume the physical meaning of connections between panels. The hold-down axial stiffness is also assigned at the interface to the lines and then, in the *xlam-driver* application, the extreme points of the lines are identified and the hold-down stiffness is assigned to the springs of these points.

In the “Surfaces” menu any line has an assigned connection, set as default to the property “Continuity connection”. It can be changed assigning previously created properties: these properties allow to choose a particular type of connection for a line that is a connection between two consecutive panels, which have that line in common. Clicking twice on “Connection” of the desired line, another connection property of the ones already created can be selected for that line (with reference to Figure 5.30).

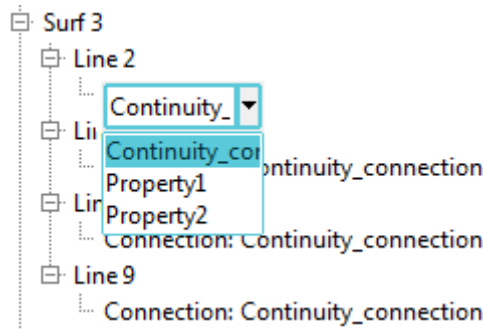


Figure 5.30 Selection of a connection property

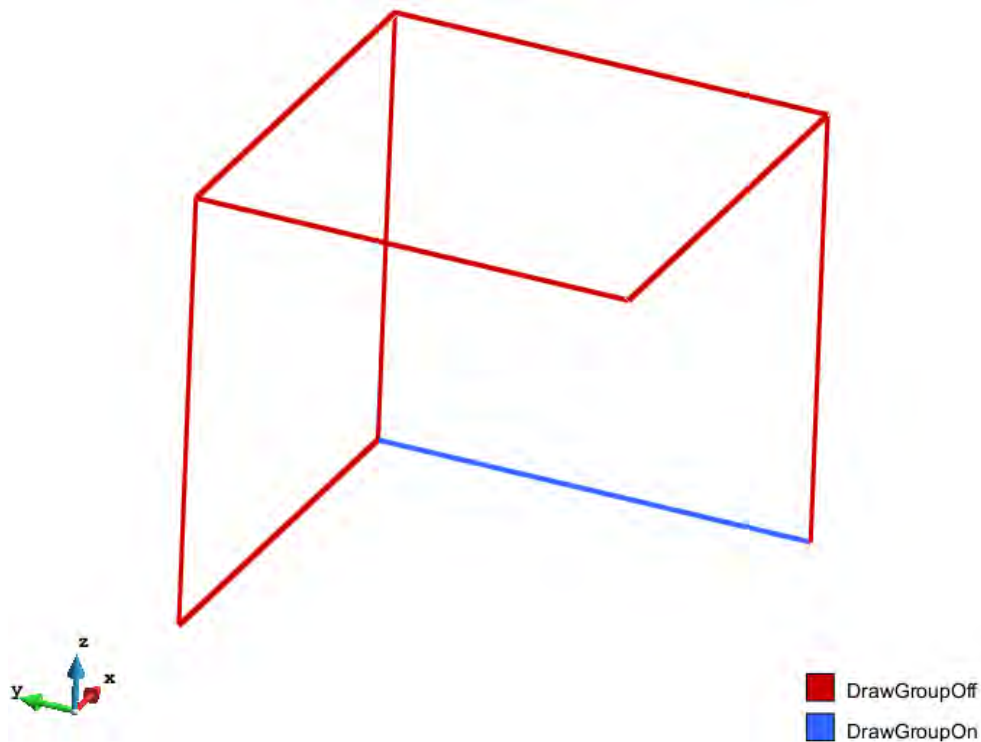


Figure 5.31 Drawing of line 2

With a more difficult geometry, it may be difficult to recognise the line to which you are assigning a property. To overcome this problem, you can click with the right

mouse button on the desired line and select "Draw": thus, the selected line will be visualised in blue colour and the other ones in red for 5 seconds, therefore allowing to easily recognise the line. For example, selecting the drawing of line 2, the geometry will appear as in Figure 5.31.

For the example analysed, Property1 will be assigned to line 7 and Property2 to line 11; the other connection properties are left with the default "Continuity connection". All the connection properties are randomly assigned because the real analysis is not the goal of this simple example. In the example of Chapter 7, they will be assigned sensibly, depending on the connection of the floor slabs and walls, assigning sensible parameters to the different connection properties as well. The "Surfaces" menu, completed by the assignation of the connection properties to the lines, is shown in Figure 5.32.

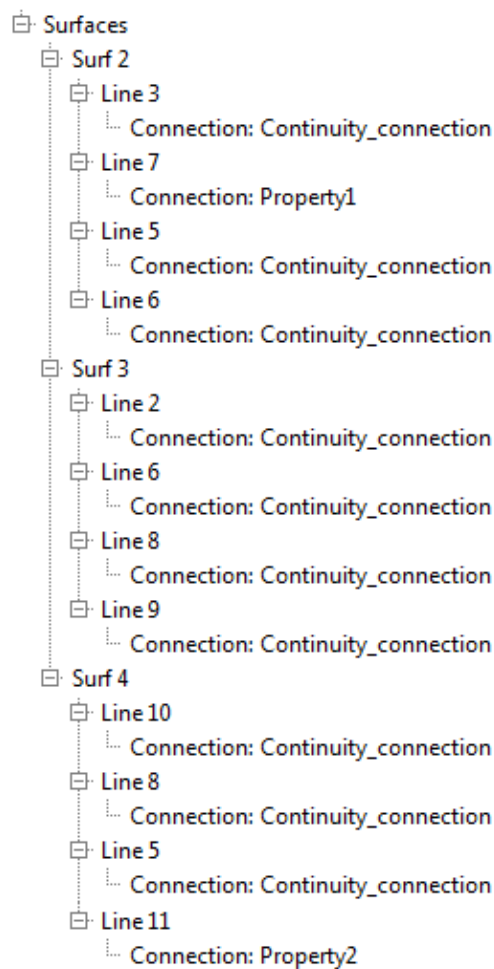


Figure 5.32 "Surfaces" menu with assigned connection properties

At this point the connection properties are completely defined and assigned to the lines, therefore other model properties can be created and assigned.

5.5.3 Elements properties creation

The elements properties can be actually created after the assignation of the material (Section 5.7), but they will be explained before, as belonging to the model properties. They can be created clicking twice on the menu “*Properties*” (see Figure 5.13), so that the window of Figure 5.33 will appear.

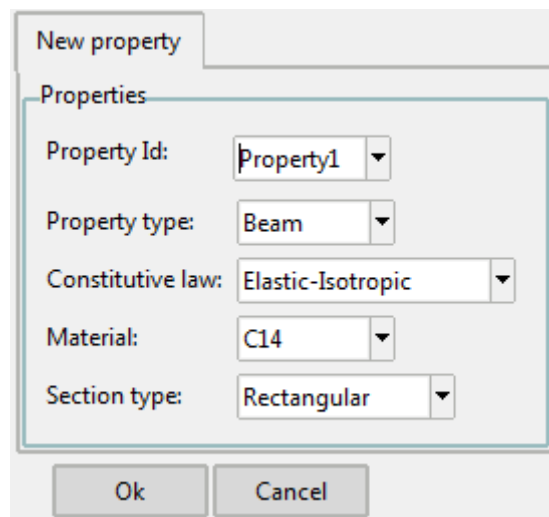
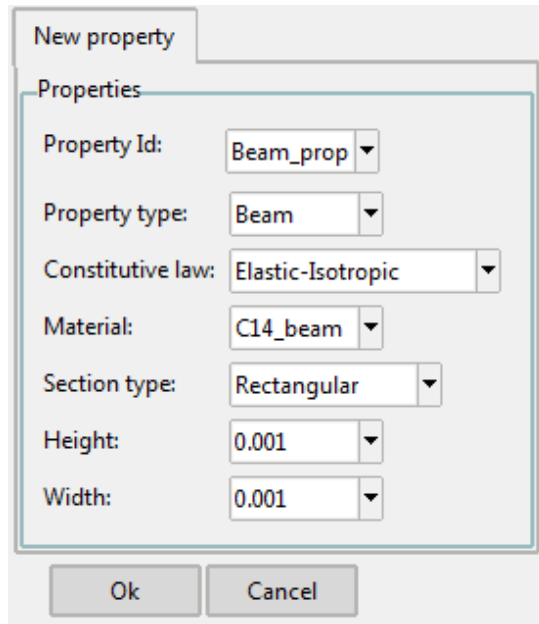


Figure 5.33 Elements “*Properties*” menu

With reference to Figure 5.33, the “*Property Id*” is the name of the new property that will be created, to whom any name can be assigned. The “*Property type*” can be “*Beam*”, “*Solid*” or “*Shell*”. The subsequent parameters change depending on the property type, so the different property types will be explained separately.

In case of beam property type, the “*Constitutive law*” can only be “*Elastic-Isotropic*”, the “*Material*” can be selected from the already default materials database, the “*Section type*” is rectangular and the “*Height*” and “*Width*” of the section should be set. For the example, a beam property named “*Beam_prop*” is created with the parameters of Figure 5.34.



The screenshot shows a 'New property' dialog box with the following parameters:

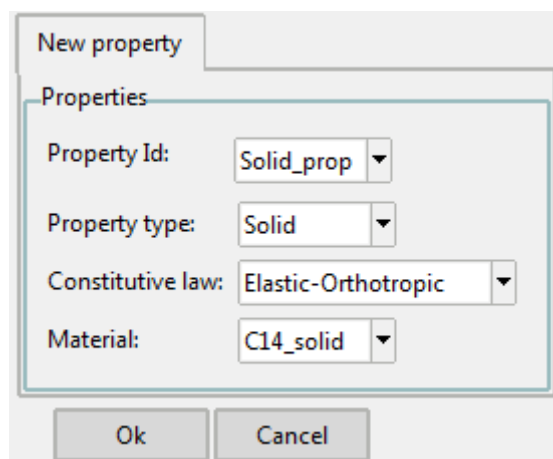
Parameter	Value
Property Id:	Beam_prop
Property type:	Beam
Constitutive law:	Elastic-Isotropic
Material:	C14_beam
Section type:	Rectangular
Height:	0.001
Width:	0.001

Buttons: Ok, Cancel

Figure 5.34 Beam property parameters

Again in this case, height and width of the section are set with random values, because the goal is not the analysis. Relatively to the material, currently it can be considered just a name which can be selected from the already default materials database; it will be explicated in Section 5.7.

In case of solid property type, the “*Constitutive law*” can be “*Elastic-Isotropic*” or “*Elastic-Orthotropic*”; if later you want to use an orthotropic shell, you should select the Elastic-Orthotropic constitutive law for the solid as well. For this example, a solid property named “*Solid_prop*” is created with the parameters of Figure 5.35.



The screenshot shows a 'New property' dialog box with the following parameters:

Parameter	Value
Property Id:	Solid_prop
Property type:	Solid
Constitutive law:	Elastic-Orthotropic
Material:	C14_solid

Buttons: Ok, Cancel

Figure 5.35 Solid property parameters

Relatively to the material, similarly to the beam property, it can be selected from the already default materials database; currently it can be considered just a name, it will be explicated in Section 5.7.

In case of shell property type, the “*Constitutive law*” can be “*Elastic-Isotropic*” or “*Elastic-Orthotropic*”, however, according to the assumption that will be explained in Section 4.2, an orthotropic shell will be used. For this example, a shell property named “*Shell_prop*” is created with the parameters of Figure 5.36.

The image shows a 'New property' dialog box with the following parameters:

Parameter	Value
Property Id:	Shell_prop
Property type:	Shell
Constitutive law:	Elastic-Orthotropic
Material:	C14_shell
Thickness:	1.0
Angle:	0
Number of layers:	5
Thickness layer 1:	0.2
Thickness layer 2:	0.2
Thickness layer 3:	0.2
Thickness layer 4:	0.2
Thickness layer 5:	0.2
Thickness layer 6:	0.0
Thickness layer 7:	0.0
Thickness layer 8:	0.0
Thickness layer 9:	0.0
Solid property:	Solid_prop

Buttons: Ok, Cancel

Figure 5.36 *Shell property parameters*

Relatively to the material, similarly to the other elements properties, it can be selected from the already default materials database; currently it can be considered just a name, then it will be explicated in Section 5.7. The other parameters that should be set in case of shell property type are:

- Thickness: this is the total thickness of the panel, in m.
- Angle: this is the angle of grain orientation of the outer layer of the panel, in degrees. It is assumed to be equal to zero if the grain of the outer layer is arranged according to the 1-axis of the shell (local x-axis), otherwise it should be set equal to ninety degrees (see Section 4.2).
- Number of layers: this is the number of layers of the panel, which can be equal to 3, 5, 7 or 9. It is not possible to set values different from these ones because the modelling of the composite shell is only predisposed for these values (see Section 6.4.8).
- Thickness layer i : this is the thickness of the i^{th} layer, in m. The first layer is the outer one; being the number of layers always odd, the two sides of the panel can be interchangeably used as reference for the outer layer.
- Solid property: this is the name of the solid related to the shell. To any shell defined with a special material, a solid is related; thus enabling the creation of the composite shell and the assignation of the material properties to it (see Section 6.4.8).

It is noticeable that the three element properties have been created with three different materials. Actually, the material is always the same, it is timber C14; nonetheless, any element property needs a different material, even if their parameters are equal. This is because the beam uses an Elastic-Isotropic constitutive law, while the other two elements use an Elastic-Orthotropic constitutive law. Moreover, the solid and the shell, despite using the same constitutive law, need to be characterised by different materials, to enable the assignation of the material properties to the whole panel and its different layers (see Section 6.4.8).

Each of the element properties parameters explained presents, on the interface, an abbreviated name that may be not perfectly clear, for a matter of space at the interface.

Anyway, placing the mouse over the box corresponding to each of the parameters, pop-ups that explain their meaning will appear.

At the end of this step the elements “*Properties*” menu results as in Figure 5.37.

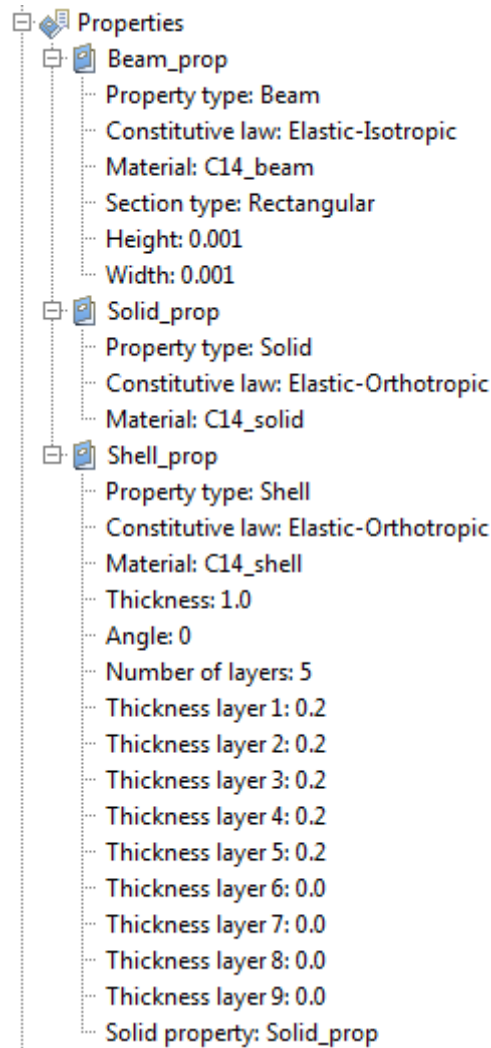


Figure 5.37 Elements “*Properties*” end menu

In this explanatory example, only one property per element has been created; in a real example it may be necessary to define more than one property per element, to assign, for instance, a different beam section or different layer thicknesses to the shells.

5.5.4 Elements properties assignation

Once the elements properties have been created, they should be assigned to the elements; the “*Elements*” menu is shown in Figure 5.38.

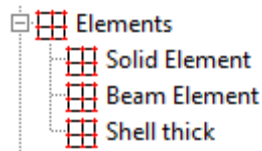


Figure 5.38 “*Elements*” menu

Firstly, the property can be assigned to the beams: clicking twice on “*Beam Element*” a new group for the beams can be created, with the property “*Beam_prop*”; this will be assigned to all the lines (reference is made to Figure 5.39).

Similarly, the property can be assigned to the shells clicking twice on “*Shell thick*”, so that a new group for the shells is created, with the property “*Shell_prop*”; this will be assigned to all the surfaces (reference is made to Figure 5.40).

Finally, clicking twice on “*Solid Element*”, a new group of solid can be created, with the property “*Solid_prop*” (reference is made to Figure 5.41). Importantly, this will not be assigned to any geometric entity because they are not solid elements; it is only created to give the properties of the material to the different layers of the shell (Section 6.4.8).

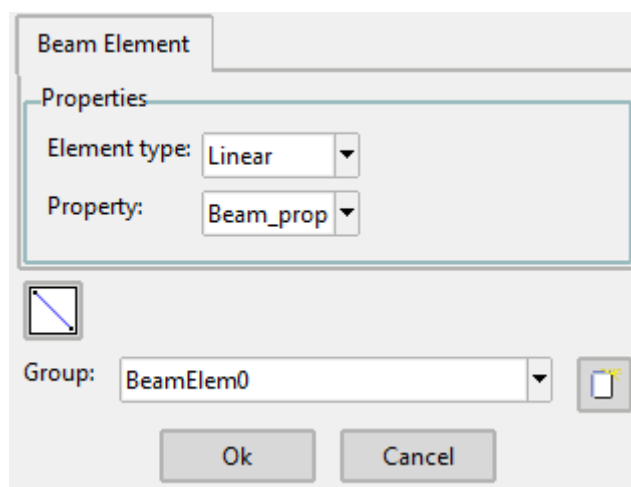


Figure 5.39 *Beam element assignation*

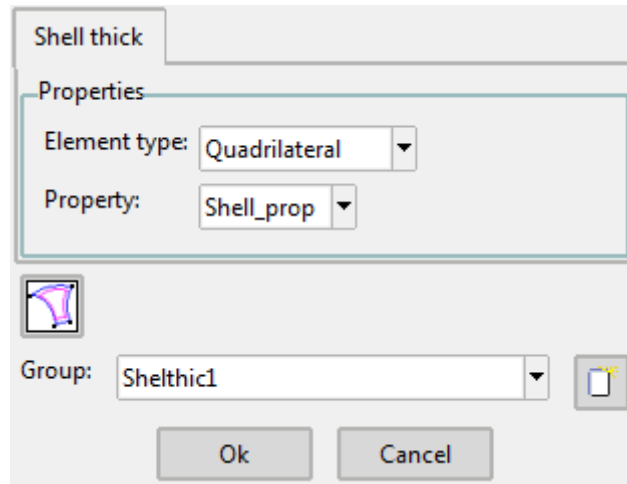


Figure 5.40 *Shell thick element assignment*

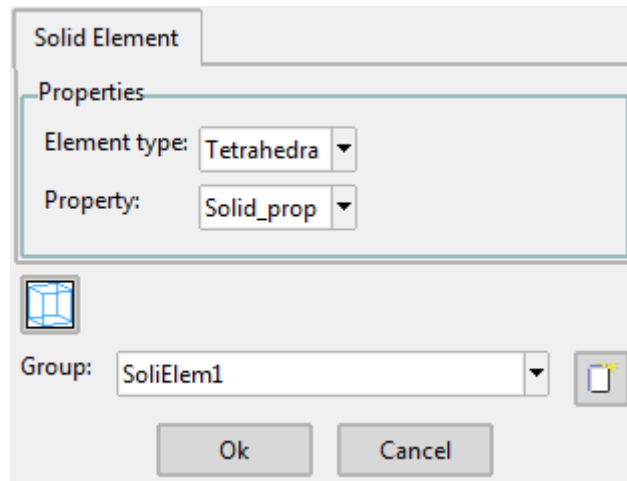


Figure 5.41 *Solid element assignment*

The order of assignment of the properties to the elements does not matter: the shells can be assigned before the beams and this would not constitute a problem.

As already mentioned, one property per element has been created, so any different element property is assigned to all the geometric entities to which it refers. Differently, in a real example, any property can be assigned to the desired entities, not necessarily to all the same geometric entities. The only restriction is to assign only one property to any geometric entity, because it is not permitted, for instance, that a beam has two different materials or two different section heights.

At the end of this step the “*Elements*” menu appears as shown in Figure 5.42.

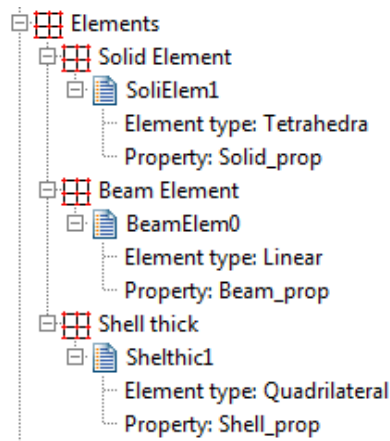


Figure 5.42 “Elements” end menu

5.5.5 Loads and boundary conditions assignation

The assignation of loads and boundary conditions is the same of any GiD problem type. Hence, this tutorial will not present it too extensively; its description will be limited to showing (Figure 5.43) and briefly explaining the menu.

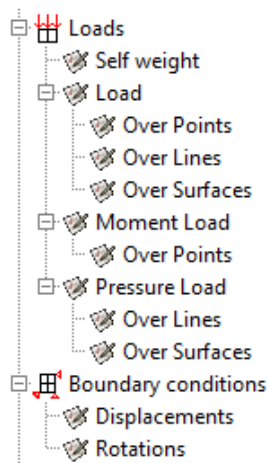


Figure 5.43 “Loads” and “Boundary conditions” menu

The “Loads” menu contains different types of loads that can be assigned: the self-weight, a generic load, a moment load and a pressure load.

The self-weight can be assigned to points, lines, surfaces and volumes: clicking twice on it, there is the possibility to choose one of these geometrical entities and then assign

the load to the type of selected entity (remember that it is set so that the gravity is on – z).

The generic load can be assigned to points, lines and surfaces: there are three sub menus which allow to assign a different value of force in x, y and z direction to each type of entity.

The moment load can be only assigned to points: there is a sub menu which allows to assign a different value of moment in x, y and z direction to the points.

The pressure load can be assigned to lines and surfaces: there are two sub menus that allow to assign the direction and the value of the pressure load to the type of selected entity.

The “*Boundary conditions*” menu contains different types of conditions that can be assigned: displacements and rotations.

The displacements can be assigned to points, lines, surfaces and volumes: clicking twice on it, there is the possibility to choose one of these geometrical entities and then assign the desired value to the type of selected entity.

The rotations can be assigned to points, lines and surfaces: clicking twice on it, there is the possibility to choose one of these geometrical entities and then assign the desired value to the type of selected entity.

For the example described, nor loads nor boundary conditions will be assigned; since they are not features of the new problem type, they are not interesting in this case of trivial example not aimed to analysis.

5.5.6 Results menu

The “*Results*” menu is the same of any GiD problem type; therefore, it will not be outlined too extensively in this tutorial. The menu will be only shown (Figure 5.44) and briefly explained.

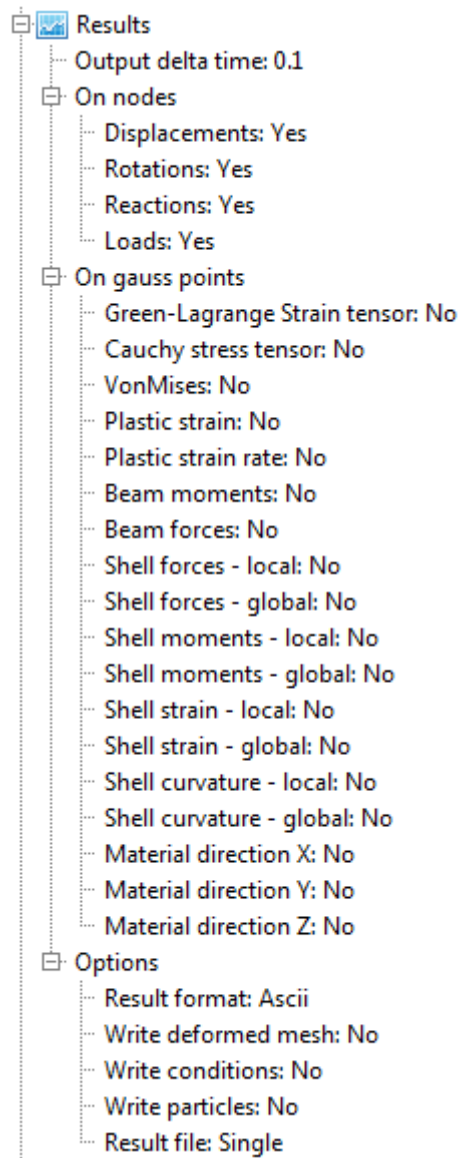


Figure 5.44 “Results” menu

The “Results” menu contains the output delta time, the results that will be shown in the post-process and a number of options for the results format; each one of the options can be changed at will.

5.6 Group properties definition

The first icon of the *xlam-kratos* menu enables to define the group properties; clicking on it, the window of Figure 5.45 will appear.

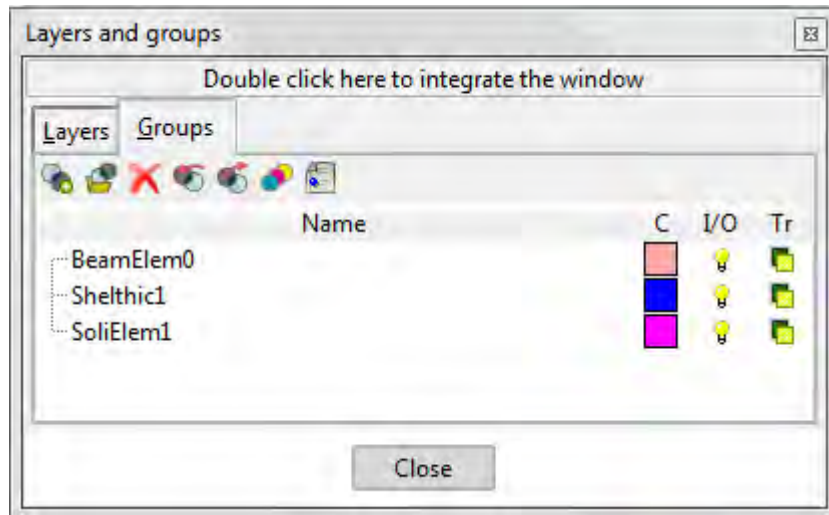


Figure 5.45 “Groups” menu

The groups that appear in this menu are the ones just created by the definition of the elements properties. Other groups can be created directly from this menu: it is another possibility to define the elements properties; then they should be assigned to the elements.

Similarly, in case of creation of loads or boundary conditions groups, they will appear in this window (Figure 5.45). Again, other loads or boundary conditions groups can be created directly from this menu; then they should be assigned to the elements.

5.7 Material properties definition

The third icon of the *xlam-kratos* menu enables to define the material properties; clicking on it, the window of Figure 5.46 will appear.

As Figure 5.46 shows, the “*Materials*” menu contains a database of already default materials: these are the different types of timber classified by the code EN 338: 2004 (see Section 2.3).

Any material presents a sub-menu (Figure 5.47) in which the values of its main properties are defined; this will be not explained in detail because it is almost the same of the code table (Section 2.3).

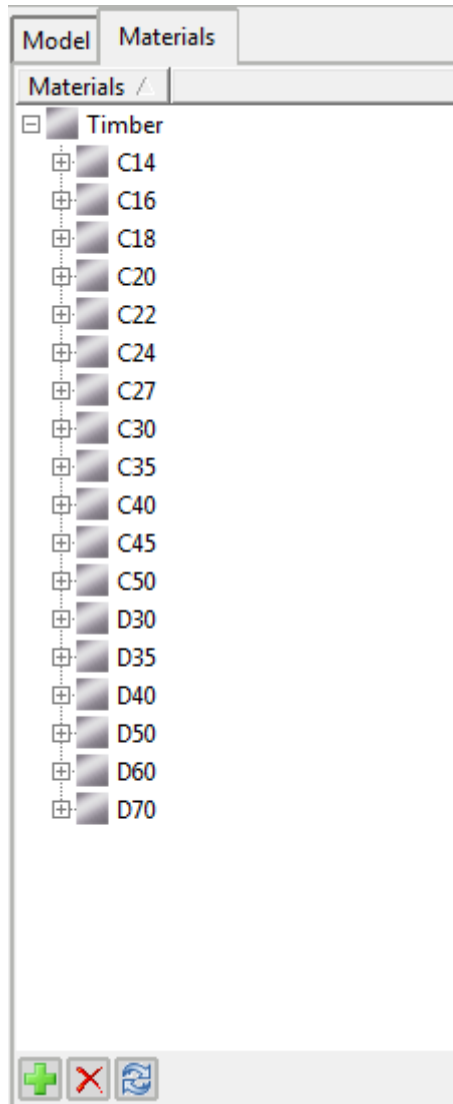


Figure 5.46 “Materials” menu

With reference to Figure 5.47, in the first section there is just a description of the material with its density; then it is sub-classified in “*Isotropic*” and “*Orthotropic*”.

Even if the X-Lam panels are made by an orthotropic material, someone might prefer to model them with an isotropic material; this is the reason why the material menu is predefined with isotropic material too. In this case, only two values are set, the Young modulus and the Poisson ratio. The former is set equal to E_0 for any material type; the latter is set to zero because of the particular mode of the panels manufacture: as a matter of fact the tables are not perfectly disposed side by side and glued the ones with each other longitudinally on the short side.

The “*Orthotropic*” section is divided into different sections: the first one requires the three Young modulus and the three Poisson ratios, the second one requires some strength values and the last section requires the characteristic and mean values of the density. For any default material, all the values of its properties are already set, by means of the criteria mentioned in Sections 2.3 and 4.2.

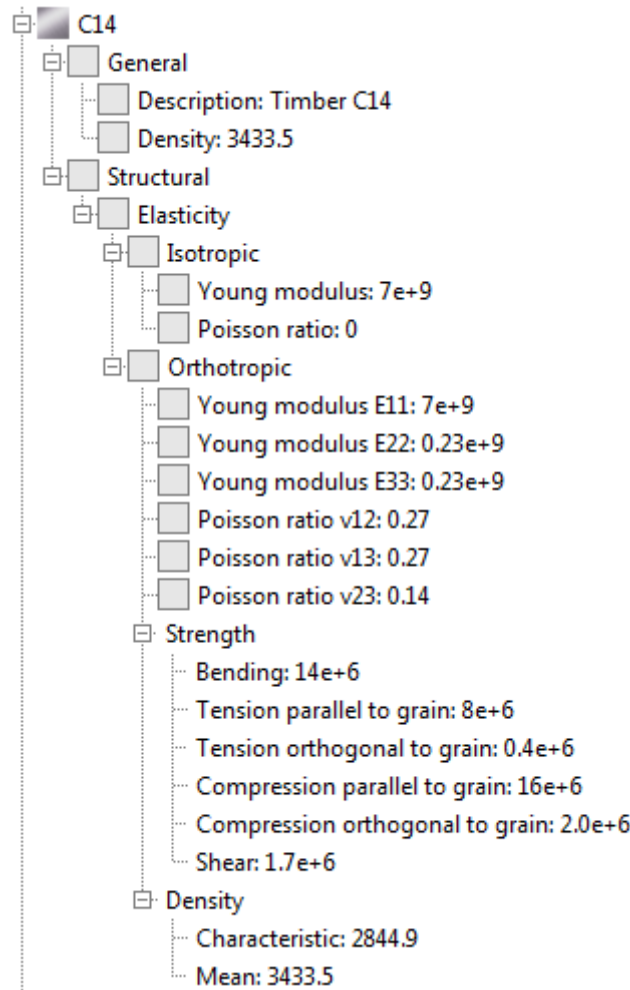


Figure 5.47 *Material sub-menu*

Selecting an existing material and clicking the right mouse button, the window of Figure 5.48 will appear. This window enables to create a new material, delete, rename and copy it. Selecting “*New Material*”, a new material can be created and all the values of its properties should be set. Because of this, if you want to create a new material with the same properties of the one selected, it is more convenient to use the command “*Copy material*”, which indeed creates a new material with the same properties values

of the previous one. This command is particularly useful when you want to use the same material for all the element types, but you have to define a different one for each one of them (see Section 5.5.3). Therefore, any material can also be renamed or deleted using the relative command.

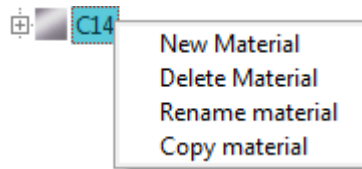


Figure 5.48 Material creation, removal and renaming window

For the example analysed, three different materials have been created copying the “*C14*”, and renaming them as “*C14_beam*”, “*C14_solid*” and “*C14_shell*”. These are the three materials selected in the definition of the elements properties; importantly, the materials should be created earlier, otherwise they will not appear in the list of materials of the elements properties.

5.8 Geometry meshing

Once all the properties have been defined, lines and surfaces should be meshed in order to launch the calculation. In general, the type of mesh can be selected from its menu, but, in this case, a mesh with quadrilateral element type is set as default, this is because the thick shell only works with linear quadrilateral mesh. As default, it is also set that the lines are meshed too, while in general this would be selected from the mesh criteria menu.

With all these simplifications, the only thing to decide is if using a structured or unstructured mesh. As default, GiD sets an unstructured mesh. This implicates that, selecting from the top command line “*Mesh - Generate mesh*” or using the shortcut Ctrl-g, an unstructured mesh will be generated and a window for the selection of the mesh size (Figure 5.49) will appear.

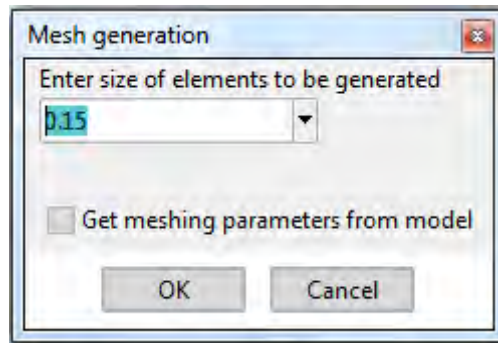


Figure 5.49 “Mesh generation” window

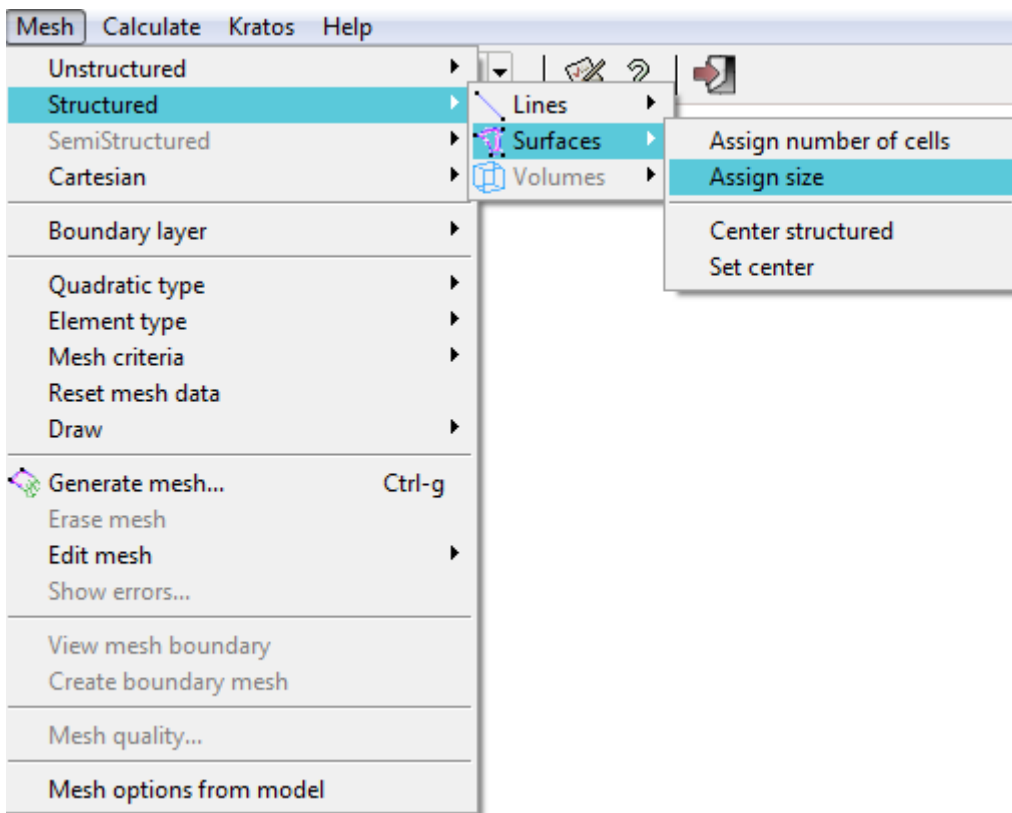


Figure 5.50 Structured mesh selection

To use a structured mesh, it can be selected from the top command line “*Mesh – Structured – Surfaces*” (Figure 5.50), and then size or number of cells can be set.

Selecting, for example, “*Assign size*”, the same or a different mesh size can be assigned to each surface; then, the mesh can be generated and the window of Figure 5.49 will appear. In this case, it is important to select “*Get meshing parameters from the model*”

(with reference to Figure 5.49) because the mesh is not a default unstructured one, but a structured mesh with the assigned size.

For the example, a mesh size of 0.5 m will be used, which is not a refined mesh, as the goal is not the calculation of the x-lam panels, but only to show how the pre-processor works. This will be more easily understood if fewer elements are present. In general, the mesh size depends on how much detailed the analysis will be. The resulting geometry mesh, showing also the GiD ID labels of lines and elements, is shown in Figure 5.51.

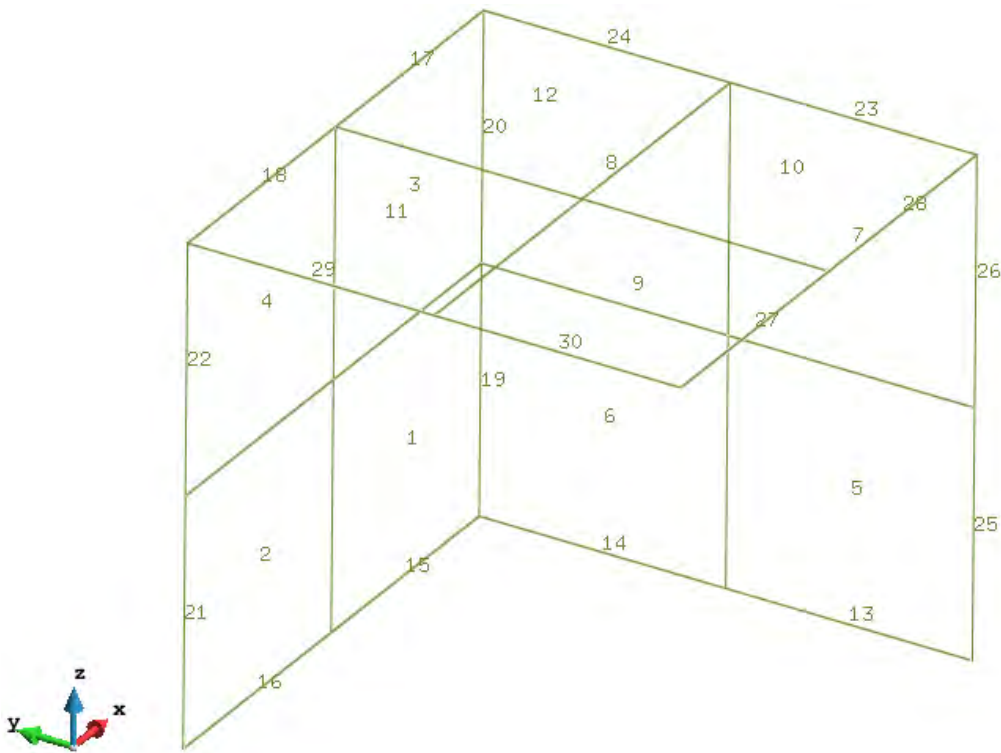


Figure 5.51 Meshed geometry

5.9 File saving

The calculation cannot be launched if the file is not saved, so, if it has not been done before, at this point it will be necessary to save it. As for any ordinary program, using the command “*File - Save as*”, the file can be saved in the desired folder.

When a file is saved, the processor generates a folder containing the different files described in Section 3.1.1, which all have the same name. This means that, if you want to change the name of the folder when it has been saved, it cannot be done simply renaming the folder, but all the within files must be renamed.

5.10 Calculation

Once the file has been saved, the calculation can be launched using the command “*Calculate*” of the top command line or the shortcut F5.

CHAPTER 6

PRE-PROCESSOR PROGRAMMING DETAIL

The problem type *xlam-kratos* has been described in Chapter 5 only at interface level, providing a user tutorial. It is clear that, when it has been created, the tools were not just available and a TCL code has been written in order to obtain the necessary features. As a matter of fact, to create a new problem type, it is necessary to explain to the processor what we need, and this should be done by means of a programming code. This is the real work in the creation of a new problem type because, without this phase, it is not possible to have a graphical feedback. Moreover, this part is the more difficult yet interesting one in the project of the pre-process: the interface is only the result of the code, whilst the code is the real work.

The interface has been described previously (in Chapter 5), in order to better understand which were the purposes when programming the TCL code. Having seen the interface, it is now easier to understand what we are talking about, describing the tools of the code. The following sections will focus on the main features of the code, and how the interface menu and the desired data have been obtained.

6.1 Surface creation and removal from the menu tree

The “*Surfaces*” menu is a section of the tree that belongs to both the modes of connection properties, the custom and the standard one; it enables to assign the connection properties to each line, as already described in Section 5.5.2. The menu is at first empty, then, as the user draws the geometry, it goes growing.

Any time the user draws a surface, its identification number appears in the tree with all the lines belonging to it; the lines are also shown with their ID. In the same way, when a surface is removed by the user, the surface and its belonging lines are removed from the tree.

6.2 Line drawing

In order to easily recognise a line to which you are assigning a connection property, the line can be drawn with a different colour respect to all the others (see Section 5.5.2). This feature has been obtained creating two hidden groups with different colours: a group contains only the selected line that the user want to recognise, the other one contains all the other lines. The two groups only persist for 5 seconds in the “Groups” menu, then they are automatically removed, to allow identification of other lines.

Figure 6.1 shows the section of the code which enables the drawing of the selected line.

```
proc ::xlam::DrawLine {nombre} {
... set id [lindex $nombre end]
...
... catch {[GiD_Groups delete "DrawGroupOn"]}
... catch {[GiD_Groups delete "DrawGroupOff"]}
... GiD_Groups create "DrawGroupOn"
... GiD_Groups create "DrawGroupOff"
...
... GiD_Groups edit state DrawGroupOn "hidden"
... GiD_Groups edit state DrawGroupOff "hidden"
...
... GiD_Groups edit color DrawGroupOn #3366ffff
... GiD_Groups edit color DrawGroupOff #cc0000ff
...
... GiD_EntitiesGroups assign DrawGroupOn lines $id
...
... set assignedlines [list]
...
... GiD_EntitiesGroups assign DrawGroupOff lines [GiD_Geometry list line 1:end]
... GiD_EntitiesGroups unassign DrawGroupOff lines $id
...
...
... GiD_Groups draw "DrawGroupOn DrawGroupOff"
... GiD_Redraw
... after 5000 {
... .. catch {[GiD_Groups end_draw]}
... .. catch {[GiD_Groups delete "DrawGroupOn"]}
... .. catch {[GiD_Groups delete "DrawGroupOff"]}
... .. GiD_Redraw
... }
}
```

Figure 6.1 “Draw line” section of the code

The two groups are hidden to the user, but, currently, they will be made visible to show how they work. Considering the example of Chapter 5, especially Section 5.5.2, line 2 can be drawn. Two groups will be created, “DrawgroupOn” and

“*DrawgroupOff*”: the former contains only line 2, which is the selected one; the latter contains all the other lines.

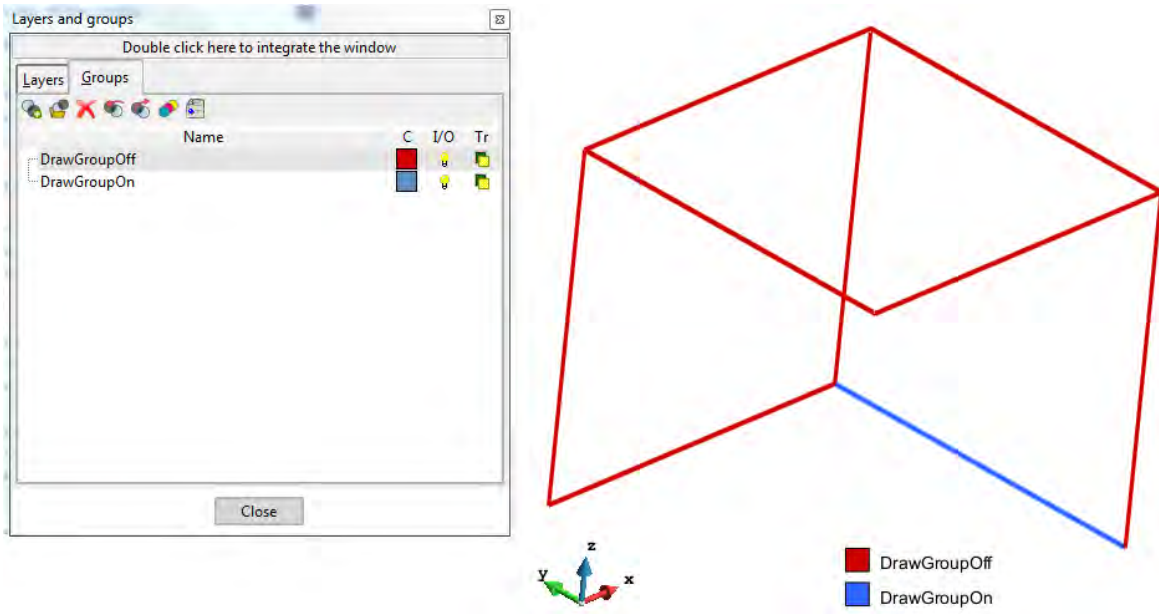


Figure 6.2 “Draw line” groups and geometry

Figure 6.2 shows the “*Groups*” menu, in which the two drawing groups appear, and the geometry, coloured according to the colours of the groups.

6.3 New IDs definition

A fundamental issue is the description of the procedure implemented in the code about the definition of the new surfaces and lines IDs; they are changed because of computational reasons of the *xlam-driver* application.

Relatively to the surfaces, it is necessary that all the identification numbers are subsequent. This is in general true if the user does not delete any surface, but, if he does, as seen in Section 5.5.2, this surface is removed from the tree and the other ones maintain the same identification number. To avoid problems in case of removal of a surface, but to leave the user the possibility of deleting it, without having to redraw the whole geometry, the surfaces IDs are renumbered. Renumbering consists in removing

the possible jumps of numbers and assigning to the surfaces a number from one up to the total number of surfaces of the final geometry.

Relatively to the lines, it is necessary that all the identification numbers are subsequent between them, and that they are subsequent to the surfaces ones too. As for the surfaces, when a line is deleted, it is removed from the tree. As mentioned, it is noticeable that GiD has a hierarchy of entities, which imposes that lines cannot be removed if firstly all the surfaces containing them have not been removed. This means that, if a line is removed, also all the surfaces to which it belongs should have been previously removed from the drawing and, so, from the tree. To avoid problems in case of removal of a line, but to leave the user the possibility of deleting it, without having to redraw the whole geometry, the lines IDs are renumbered. Renumbering consists in removing the possible jumps of numbers and assigning to the lines a number that, differently from the surfaces, does not start from one. Due to computational reasons, the second criteria of renumbering the lines IDs is that they start from the number next to which the surfaces end, continuing up to their end.

6.4 Files writing

Many different files are written by the code, some of them useful for *xlam-driver* application and others useful for the planned post-process. Because of the creation of these files in the code, when the calculation is launched, the processor automatically generates the files that can be used as data by the *xlam-driver* or by the post-process.

In the following sections, all these files will be analysed, in order to understand what exactly the processor is doing with the information given by the user, when he draws the geometry and assigns the elements and connection properties. Additionally, the files associated to the example of Chapter 5 will be shown, to better understand what they provide graphically.

6.4.1 Geometry info

The file “*geometry-info*” provides a matrix, necessary for *xlam-driver* application, which relates old and new ID of surfaces and lines. After renumbering the surfaces and lines with the criteria described in Section 6.3, a file showing the list of the old and new ID of surfaces and lines is written.

For the example described in Chapter 5, the file “*geometry-info*” results as shown in Table 6.1.

Table 6.1 “*Geometry-info*” file

New surfaces ID	Old surfaces ID
1	2
2	3
3	4
New lines ID	Old lines ID
4	2
5	3
6	5
7	6
8	7
9	8
10	10
11	9
12	11

In order to understand the changes about the surfaces, the first three rows of Table 6.1 can be analysed. The first surface has been removed before the assignation of the connection properties to the lines (see Section 5.5.2), so the list of old surfaces IDs, which are the ones defined by default by GiD, does not contain the ID number one. To restore this jump in number, for which the surfaces begin directly from the number two, the surfaces IDs are renumbered starting from one up to three, which is the total number of surfaces. It is clear that if no surface had been deleted, the new surfaces IDs would coincide with the old ones.

Relatively to the changes about the lines, lines 1 and 4 have been removed from the geometry (see Section 5.5.2), so the list of old lines IDs, which are the ones defined by default by GiD, does not contain the IDs 1 and 4. Moreover, the list of old lines IDs starts from the number two. The lines IDs are renumbered starting from four because the surfaces are three, and all the numbers are subsequent up to twelve, that is equal to the sum of the number of surfaces and the number of lines. Notably, even if no line had been deleted, the new lines IDs would still be different from the originals, because they would start from the total number of surfaces.

6.4.2 Connection info

For any surface, the lines belonging to them are known and listed in the tree of the “*Surfaces*” menu; accordingly, the connection properties of the lines, assigned by the user, are known and listed in the tree.

The file “*connection-info*” provides a matrix, necessary for the *xlam-driver* application, which shows the list of the new lines IDs, the new surfaces IDs to which the lines belong, and their stiffness connection properties. For the example described in Chapter 5, the file “*connection-info*” results as shown in Table 6.2.

It is noticeable that the lines belonging to more than one surface appear in the table (Table 6.2) more times. For example, line 6 (former line 5) appears as belonging to surfaces 1 and 3 (respectively former surfaces 2 and 4). This is foregone if you remember that the user assigns to each line belonging to each surface a connection property; therefore, if a line belongs to more than one surface, he assigns more connection properties, which in general will be different.

The third column (reference is made to Table 6.2) shows the whole stiffness of the connection property, meaning eight stiffness, not only the ones set by the user (or resulting from the parameters set by the user). In addition to the five stiffness visible at the interface, the last three values exhibit the rotational stiffness, which are always set to a small value equal to 100 N/m. In case of “*Continuity connection*” property, all the

stiffness are set to a very high value equal to 10^{12} N/m; the only two lines that do not have this connection property, exhibit different stiffness. Line 8 (former line 7) is the one to which Property1 has been assigned (see Section 5.5.2). The property was defined by means of the “*Stiffness*” mode, so it shows the stiffness as they have been set by the user. Line 12 (former line 11) is the one to which Property2 has been assigned. The property was defined by means of the “*Connection parameters*” mode, so the hold-down axial stiffness and the shear stiffness in the plane of the panel are calculated by the processor as explained in Section 2.7.2, while the others stiffness are set to 100 N/m.

Table 6.2 “*Connection-info*” file

New lines ID	New surfaces ID	Stiffness connection property			
4	2	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
5	1	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
6	1	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
6	3	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
7	1	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
7	2	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
8	1	1,0E+07	1,0E+07	100,00	1,0E+07
		100,00	100,00	100,00	100,00
9	2	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
9	3	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
10	3	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
11	2	1,0E+12	1,0E+12	1,0E+12	1,0E+12
		1,0E+12	1,0E+12	1,0E+12	1,0E+12
12	3	79398089,22	79398089,22	100,00	7939808,92
		100,00	100,00	100,00	100,00

6.4.3 Surface element info

The file “*surf-elem-info*” provides a matrix, necessary for the *xlam-driver* application, which shows the list of the identification numbers of the mesh elements, associated to the ID (the new one) of the surface to which they belong. For the example described in Chapter 5, the file “*surf-elem-info*” results as shown in Table 6.3.

Table 6.3 “*Surface-element-info*” file

Mesh surfaces elements	New surfaces ID
1	1
2	1
3	1
4	1
5	2
6	2
7	2
8	2
9	3
10	3
11	3
12	3

This file enables to know the identification number of the surfaces to which the elements of the mesh belongs, in order not to lose the information about them. It is obtained in the code defining an occult group that in fact refers the mesh elements to the original elements of the geometry.

6.4.4 Line element info

The file “*line-elem-info*” provides a matrix, necessary for the *xlam-driver* application, which shows the list of the identification numbers of the mesh elements, associated to the ID (the new one) of the line to which they belong. For the example described in Chapter 5, the file “*line-elem-info*” results as shown in Table 6.4.

Table 6.4 “Line-element-info” file

Mesh lines elements	New lines ID
13	4
14	4
15	5
16	5
17	6
18	6
19	7
20	7
21	8
22	8
23	9
24	9
25	11
26	11
27	10
28	10
29	12
30	12

Similarly to the “*surf-elem-info*”, this file enables to know the identification number of the lines to which the elements of the mesh belongs, in order not to lose the information about them. It is obtained in the code defining an occult group that in fact refers the mesh elements to the original lines of the geometry.

6.4.5 More connections

The file “*More-Connections*” provides a matrix, necessary for the post-process, which shows the list of the new lines IDs, the new surfaces IDs to which the lines belongs and their resistance connection properties. For the example described in Chapter 5, the file “*More-Connections*” results as shown in Table 6.5.

It is noticeable that the lines belonging to more than one surface appear in the table (Table 6.5) more times. For example, line 6 (former line 5) appears as belonging to

surfaces 1 and 3 (respectively former surfaces 2 and 4). This is foregone if you remember that the user assigns to each line belonging to each surface a connection property; therefore, if a line belongs to more than one surface, he assigns more connection properties, which in general will be different.

The third column (reference is made to Table 6.5) shows the whole resistance of the connection property, meaning three design load-carrying capacities (one for any hold-down and one for the brackets or distributed nailing), calculated by means of the three characteristic load-carrying capacities set by the user. In case of “*Continuity connection*” property, nothing is set because this property means that there is no connection on the line, it is ensured the continuity. The only two lines that do not have this connection property, exhibit the three values of the capacities, calculated by the processor as explained in Section 2.7.3.

Table 6.5 “*More-Connections*” file

New lines ID	New surfaces ID	Resistance connections property
4	2	
5	1	
6	1	
6	3	
7	1	
7	2	
8	1	84615.38 84615.38 59230.77
9	2	
9	3	
10	3	
11	2	
12	3	50769.23 50769.23 10153.85

6.4.6 More materials

The file “*More-Materials*” provides some information about the material, necessary for the post-process: it shows for any material, assigned to an element property, the strength and density values belonging to the orthotropic material. For the

example described in Chapter 5, relatively to the shell material, the file “*More-Materials*” results as shown in Figure 6.3.

Begin Mat C14_shell	
Bending	14e+6
Tensionpar	8e+6
Tensionort	0.4e+6
Compressionpar	16e+6
Compressionort	2.0e+6
Shear	1.7e+6
Characteristic	2844.9
Mean	3433.5
End Mat	

Figure 6.3 “*More-Materials*” file

The first six lines (Figure 6.3) show the strengths:

- bending resistance;
- tension parallel to the grain;
- tension orthogonal to the grain;
- compression parallel to the grain;
- compression orthogonal to the grain;
- shear resistance.

The last two lines (Figure 6.3) show the density values:

- characteristic density;
- mean density.

These are the values set by default because the material selected is the C14, so it is one of the already defined materials; if the user defines another material with different strength and density values, the values shown in the file “*More-Materials*” will be the ones inserted.

6.4.7 Data orthotropic

The file “*data-Ortho*” provides a matrix, necessary for the calculation, which shows some information about the shell elements properties: it shows, for any shell element, assigned at least to one surface, almost all the values relative to the shell. For the example described in Chapter 5, having been defined only one shell thick element, the matrix is converted into a vector and the file “*data-Ortho*” results as shown in Table 6.6.

Table 6.6 “*Data-Orthotropic*” file

Shell ID	3
Solid ID	1
Angle	0
N° layers	5
Thickness 1	0,2
Thickness 2	0,2
Thickness 3	0,2
Thickness 4	0,2
Thickness 5	0,2
Thickness 6	0,0
Thickness 7	0,0
Thickness 8	0,0
Thickness 9	0,0

Table 6.6 shows the “*data-Ortho*” file as a vector of one column, but it is actually printed as its transposed (a vector of one row). The first row is the shell ID, which is a way to identify each element with a number instead of the name given by the user: GiD assigns as default a number to any element property defined. The second row is the solid ID, which, as the shell ID, is assigned by default by GiD instead of the name given by the user. The subsequent rows are values assigned by the user during the definition of the shell element property: the angle of grain orientation of the outer layer, the number of layers and the thickness of each layer.

This file is necessary for the calculation, especially for the modelling of the orthotropic shell as a composite cross section with a defined number of layers, which will be explained in Section 6.4.8.

6.4.8 Materials.py

The file “*materials.py*” is generated by Kratos when the calculation is launched and it has the python format. In general, it shows, for any element property, its identification number, the material assigned, the constitutive law used and, for the beam, also the height and width of the rectangular section. For the example described in Chapter 5, the first section of the file “*materials.py*” results as shown in Figure 6.4.

```
# Importing the Kratos Library
from KratosMultiphysics import *
from KratosMultiphysics.SolidMechanicsApplication import *
from beam_sections_python_utility import SetProperties
def AssignMaterial(Properties):
# GUI property identifier: Solid_prop
# GUI material identifier: C14_solid
    prop_id = 1;
    prop = Properties[prop_id]
    mat = LinearElasticOrthotropic3DLaw();
    prop.SetValue(CONSTITUTIVE_LAW, mat.Clone());
# GUI property identifier: Beam_prop
# GUI material identifier: C14_beam
    prop_id = 2;
    prop = Properties[prop_id]
    mat = LinearElasticPlaneStrain2DLaw();
    prop.SetValue(CONSTITUTIVE_LAW, mat.Clone());
    prop = SetProperties("Rectangular", [0.001, 0.001], prop);
# GUI property identifier: Shell_prop
# GUI material identifier: C14_shell
    prop_id = 3;
    prop = Properties[prop_id]
    mat = LinearElasticOrthotropic3DLaw();
    prop.SetValue(CONSTITUTIVE_LAW, mat.Clone());
```

Figure 6.4 “*Materials.py*” first section: list of elements properties

In addition to this section, another one has been implemented, which is printed inside the same file. To achieve this, a modification has been done to the TCL file which prints this one, adding it another section, written as a comment, in python format; therefore, this additional section can be read by Kratos because it is written in python language.

The additional section enables the modelling of shells with composite cross section: each layer of the composite cross section is modelled as an orthotropic material, to fit the characteristics of X-Lam panels. It will be explained below because it also

helps to understand the need of defining different materials for the shell element and the solid one.

First, it reads the file “*dataOrtho*” described in the previous paragraph and defines any column of this file as an empty vector; then, it fills the vectors with the values printed in the file “*dataOrtho*” (Figure 6.5).

```
... data_file="dataOrtho.txt"
... data1=open(data_file,"r")
... idshell=[]
... idsolid=[]
... angle=[]
... nlayers=[]
... thickness1=[]
... thickness2=[]
... thickness3=[]
... thickness4=[]
... thickness5=[]
... thickness6=[]
... thickness7=[]
... thickness8=[]
... thickness9=[]
... while True:
...     field=(data1.readline()).split()
...     if not field:
...         break
...     idshell.append(int(field[0]))
...     idsolid.append(int(field[1]))
...     angle.append(float(field[2]))
...     nlayers.append(float(field[3]))
...     thickness1.append(float(field[4]))
...     thickness2.append(float(field[5]))
...     thickness3.append(float(field[6]))
...     thickness4.append(float(field[7]))
...     thickness5.append(float(field[8]))
...     thickness6.append(float(field[9]))
...     thickness7.append(float(field[10]))
...     thickness8.append(float(field[11]))
...     thickness9.append(float(field[12]))
```

Figure 6.5 “*Materials.py*” added section: “*dataOrtho*” file reading

Thereafter, the vectors defined (Figure 6.5) are used to fill a matrix called “*data*”, as shown in Figure 6.6. After defining the matrix, the application for the modelling of the composite shell works in the following fashion. Depending on the number of layers, first the shell cross section is created; then, depending on the angle of grain orientation of the outer layer, each layer is added to the cross section with its own thickness, orientation angle, number of through-the-thickness integration points (a Simpson’s

integration rule is used) and the material property of the solid. Finally, the composite section is added to the property assigned to the shell in GiD. For the example described in Chapter 5, the panel is composed by five layers, hence the modelling is done as shown in Figure 6.7.

```

... data=Matrix(len(idshell),11)
... for i in range(len(idshell)):
...     data[i,0]=angle[i]
...     data[i,1]=nlayers[i]
...     data[i,2]=thickness1[i]
...     data[i,3]=thickness2[i]
...     data[i,4]=thickness3[i]
...     data[i,5]=thickness4[i]
...     data[i,6]=thickness5[i]
...     data[i,7]=thickness6[i]
...     data[i,8]=thickness7[i]
...     data[i,9]=thickness8[i]
...     data[i,10]=thickness9[i]
... #print(data)

```

Figure 6.6 “Materials.py” added section: matrix creation

```

... for i in range(len(idshell)):
...     if data[i,1]==5:
...         shell_section = ShellCrossSection()
...         shell_section.SetOffset(0.0)
...         shell_section.BeginStack()
...         if data[i,0]==90:
...             shell_section.AddPly(data[i,2], 90, 3, Properties[idsolid[i]])
...             shell_section.AddPly(data[i,3], 0, 3, Properties[idsolid[i]])
...             shell_section.AddPly(data[i,4], 90, 3, Properties[idsolid[i]])
...             shell_section.AddPly(data[i,5], 0, 3, Properties[idsolid[i]])
...             shell_section.AddPly(data[i,6], 90, 3, Properties[idsolid[i]])
...         else:
...             shell_section.AddPly(data[i,2], 0, 3, Properties[idsolid[i]])
...             shell_section.AddPly(data[i,3], 90, 3, Properties[idsolid[i]])
...             shell_section.AddPly(data[i,4], 0, 3, Properties[idsolid[i]])
...             shell_section.AddPly(data[i,5], 90, 3, Properties[idsolid[i]])
...             shell_section.AddPly(data[i,6], 0, 3, Properties[idsolid[i]])
...         shell_section.EndStack()
...     shell_section.AddToProperties(SHELL_CROSS_SECTION, shell_section, Properties[idshell[i]])

```

Figure 6.7 “Materials.py” added section: composite cross section creation

Moreover, the shell cross section offers the possibility of declaring an offset from the reference surface of the shell, useful to model slabs on beams when the real distance between beams and slab is not negligible. The default value is 0.0, meaning that the cross section is aligned with the shell reference surface.

With the explanation of this application, it is now clear why a solid property has been created but it has not been assigned to anything: in GiD interface only one element

property can be assigned to any geometric entity. Hence, the shell element property is assigned to the surfaces, while the solid property is defined to identify the material of the different layers of the shell. Moreover, two different materials for the solid and the shell need to be created, even if they are actually the same material, to assign the material properties of the shell to the whole panel and the material properties of the solid to the different layers.

6.5 Connections

Depending on the mode selected by the user to define the connection properties, the stiffness and load-carrying capacities are calculated or set in different ways, to be used for calculation or verification.

6.5.1 Connections stiffness

Relatively to the stiffness, calculation is done in a different way depending on the three modes of definition of the connection properties; the stiffness calculated are used for the analysis of the problem.

6.5.1.1 “Custom-Stiffness” mode

When the user defines the connection properties with the “*Custom-Stiffness*” mode, it is not necessary to calculate the stiffness because they are just set by the user. More precisely, the axial stiffness of the hold-down, the axial stiffness of the brackets or distributed nailing and the shear stiffness are set by the user, while the rotational stiffness are set as default equal to 100 N/m.

Relative to the example of Chapter 5, we can focus on line 7, then renamed as line 8 (see Section 6.4.1), to which Property1 has been assigned. Table 6.7 shows that

the stiffness displayed in the file “*connection-info*” are actually the ones defined by the user, excepting for the rotational stiffness which are set as default equal to 100 N/m.

Table 6.7 “*Connection-info*” file for line 7

New lines ID	New surfaces ID	Stiffness connection property			
8	1	1,0E+07	1,0E+07	100,00	1,0E+07
		100,00	100,00	100,00	100,00

6.5.1.2 “Custom-Parameters” mode

When the user defines the connection properties with the “*Custom-Parameters*” mode, the axial stiffness of the brackets or distributed nailing, the shear stiffness out of the plane of the panel and the three rotational stiffness are set as default equal to 100 N/m, while the other ones are calculated in two steps.

First, the slip modulus per shear plane per fastener, for timber-to-steel connections, is calculated as:

$$k_{ser} = 2 \cdot \frac{\rho_m^{1.5} \cdot d^{0.8}}{30}$$

where ρ_m is the density of the panel and d is the nail diameter of the hold-down or the bracket (or distributed nailing). Then, the axial stiffness of the hold-down and the shear stiffness in the plane of the panel are calculated as follows:

- The axial stiffness of the two hold-down:

$$K_a = k_{ser} \cdot \text{total number of nails} \cdot 2$$

The modelled stiffness is multiplied by two times the real one, because of the different behaviour, under the action of horizontal forces, of a single modelled panel compared with the same in a real situation (see Section 4.3).

- The shear stiffness in the plane of the panel (the shear stiffness of the bracket or distributed nailing):

$$K_{t1} = k_{ser} \cdot \text{total number of distributed nails}$$

Figure 6.8 shows the section of the code which calculates the stiffness to write them in the “connection-info” file.

```
set kserTension1 [expr { (($Density/9.81)**1.5) * (($TensionDiameter1*1000)**0.8)/15 } * 1000]
set kserTension2 [expr { (($Density/9.81)**1.5) * (($TensionDiameter2*1000)**0.8)/15 } * 1000]
set kserShear [expr { (($Density/9.81)**1.5) * (($ShearDiameter*1000)**0.8)/15 } * 1000]
set kaHD1 [expr { $kserTension1 * $TensionNumberNails1 * 2}]
set kaHD2 [expr { $kserTension2 * $TensionNumberNails2 * 2}]
set ka 100
set kt1 [expr { $kserShear * $ShearNumbernails}]
set kt2 100
set kb1 100
set kb2 100
set ktr 100
```

Figure 6.8 Stiffness calculation for “Custom-Parameters” mode

Relative to the example of Chapter 5, we can focus on line 11, then renamed as line 12 (see Section 6.4.1), to which Property2 has been assigned. It can be verified that the stiffness are actually correctly calculated.

The slip modulus per shear plane per fastener, for both the tension and shear connections, results:

$$k_{ser} = 2 \cdot \frac{\rho_m^{1.5} \cdot d^{0.8}}{30} = \left(2 \cdot \frac{(3433.5 \div 9.81)^{1.5} \cdot (0.004 \cdot 1000)^{0.8}}{30} \right) \cdot 1000$$

$$= 1323301.49 \text{ N/m}$$

The axial stiffness of the two hold-down results:

$$K_a = k_{ser} \cdot 30 \cdot 2 = 79398089.22 \text{ N/m}$$

The shear stiffness of the brackets or distributed nailing results:

$$K_{t1} = k_{ser} \cdot 6 = 7939808.92 \text{ N/m}$$

Table 6.8 shows that the axial stiffness of the brackets or distributed nailing, the shear stiffness out of the plane of the panel and the three rotational stiffness are set as default equal to 100 N/m, while the other ones are actually calculated correctly.

Table 6.8 “Connection-info” file for line 11

New lines ID	New surfaces ID	Stiffness connection property			
12	3	79398089,22	79398089,22	100,00	7939808,92
		100,00	100,00	100,00	100,00

6.5.1.3 “Standard” mode

When the user defines the connection properties with the “*Standard*” mode, the axial stiffness of the bracket or distributed nailing, the shear stiffness out of the plane of the panel and the three rotational stiffness are set as default equal to 100 N/m, while the other ones are calculated in two steps.

First, the slip modulus per shear plane per fastener, for timber-to-steel connection, is calculated as:

$$k_{ser} = 2 \cdot \frac{\rho_m^{1.5} \cdot d^{0.8}}{30}$$

where ρ_m is the density of the panel and d is the nail diameter of the hold-down or the bracket (or distributed nailing). Then, the axial stiffness of the hold-down and the shear stiffness in the plane of the panel are calculated as follows:

- The axial stiffness of the two hold-down:

$$K_a = k_{ser} \cdot \text{number of nails} \cdot \text{number of hold - down} \cdot 2$$

The modelled stiffness is multiplied by two times the real one, because of the different behaviour, under the action of horizontal forces, of a single modelled panel compared with the same in a real situation (see Section 4.3).

- The shear stiffness in the plane of the panel, if they are disposed brackets, is calculated as:

$$K_{t1} = k_{ser} \cdot \text{number of nails} \cdot \text{number of brackets}$$

- The shear stiffness in the plane of the panel, if it is disposed a distributed nailing, is calculated as:

$$K_{t1} = k_{ser} \cdot \text{number of nails of the distributed nailing}$$

The nail diameter and the number of nails of the hold-down and brackets are already set in the code as shown in Figures 6.9 and 6.10. The nail diameter of the distributed nailing is already set in the code as Figure 6.11 shows.

```
set TensionDiameter1 0.004
set TensionNumberNails1 20
set Rk1 31400
```

Figure 6.9 Values set in the code for the hold-down WHT340 $\Phi 4 \times 40$ with total nailing

```
set ShearDiameter 0.004
set ShearNumbernails 12
set Rk 8940
```

Figure 6.10 Values set in the code for the bracket WBR100 $\Phi 4 \times 60$ with total nailing, concrete-timber

```
set ShearDiameter 0.004
set Rk 690
```

Figure 6.11 Values set in the code for the distributed nailing with nail diameter of 4 mm and nail length of 30 mm

Figure 6.12 shows the section of the code which calculates the stiffness to write them in the “*connection-info*” file.

```
set kserTension1 [expr { (($Density/9.81)**1.5) * (($TensionDiameter1*1000)**0.8)/15 } * 1000 ]
set kserTension2 [expr { (($Density/9.81)**1.5) * (($TensionDiameter2*1000)**0.8)/15 } * 1000 ]
set kserShear [expr { (($Density/9.81)**1.5) * (($ShearDiameter*1000)**0.8)/15 } * 1000 ]
set kaHD1 [expr { $kserTension1 * $TensionNumberNails1 * 2 } ]
set kaHD2 [expr { $kserTension2 * $TensionNumberNails2 * 2 } ]
set ka 100
set kt1 [expr { $kserShear * $ShearNumbernails } ]
set kt2 100
set kb1 100
set kb2 100
set ktr 100
```

Figure 6.12 Stiffness calculation for “Standard” mode

6.5.2 Connections resistance

Relatively to the design load-carrying capacities, calculation is done in a different way depending on the three modes of definition of the connection properties; the capacities calculated will be used for verification of the connections.

6.5.2.1 “Custom-Stiffness” mode

When the user defines the connection properties with the “*Custom-Stiffness*” mode, he sets the characteristic load-carrying capacities of the hold-down and brackets or distributed nailing. With these parameters, the design load-carrying capacities are easily calculated in the following way (see Section 2.7.3):

$$R_d = k_{mod} \frac{R_k}{\gamma_M}$$

```

set retstr ""
foreach propnode [$baseNode childNodes] {
  set propid [$propnode getAttribute "id"]
  if {$propid ni [list "RkHD1" "RkHD2" "RkAB"]} {
    set propdv [$propnode getAttribute "dv"]
    append retstr "$propdv "
  } else {
    if {$propid ni [list "kmod" "gam"]} {
      set propdv [expr [$propnode getAttribute "dv"] * $kmod / $gam]
      append retstr [format "%.2f" $propdv] "
    }
  }
}
return $retstr

```

Figure 6.13 Design load-carrying capacities calculation for “*Custom-Stiffness*” mode

Figure 6.13 shows the section of the code which calculates the design load-carrying capacities of the connections to write them in the “*More-Connections*” file.

Relative to the example of Chapter 5, we can focus on line 7, then renamed as line 8, to which Property1 has been assigned. It can be verified that the design load-carrying capacities are actually correctly calculated:

$$R_{d,hold-down} = 1.1 \cdot \frac{10^5}{1.3} = 84615.38 N$$

$$R_{d,brackets/distributed\ nailing} = 1.1 \cdot \frac{7 \cdot 10^4}{1.3} = 59230.77\ N$$

Table 6.9 shows that the design load-carrying capacities of the two hold-down and the brackets are actually calculated correctly.

Table 6.9 “More-Connections” file for line 7

New lines ID	New surfaces ID	Resistance connections property		
8	1	84615.38	84615.38	59230.77

6.5.2.2 “Custom-Parameters” mode

When the user defines the connection properties with the “*Custom-Parameters*” mode, he sets the characteristic load-carrying capacity per single fastener of the hold-down and brackets or distributed nailing. With these parameters, the characteristic load-carrying capacities are calculated in the following way:

$$R_k = F_{V,Rk} \cdot \text{total number of nails}$$

So, the design load-carrying capacities are easily calculated as (see Section 2.7.3):

$$R_d = k_{mod} \frac{R_k}{\gamma_M}$$

Figure 6.14 shows the section of the code which calculates the design load-carrying capacities of the connections to write them in the “*More-Connections*” file.

```
set rk1 [expr $FvRkHD1 * $TensionNumberNails1]
set rk2 [expr $FvRkHD2 * $TensionNumberNails2]
set rk [expr $FvRkAB * $ShearNumbernails]

set rd1 [expr $rk1 * $kmod / $gam]
set rd2 [expr $rk2 * $kmod / $gam]
set rd [expr $rk * $kmod / $gam]
```

Figure 6.14 Design load-carrying capacities calculation for “*Custom-Parameters*” mode

Relative to the example of Chapter 5, we can focus on line 11, then renamed as line 12, to which Property2 has been assigned. It can be verified that the design load-carrying capacities are actually correctly calculated.

The characteristic load-carrying capacities of the hold-down and brackets or distributed nailing result:

$$R_{k,hold-down} = 2000 \cdot 30 = 60000 \text{ N}$$

$$R_{k,brackets/distributed\ nailing} = 2000 \cdot 6 = 12000 \text{ N}$$

The design load-carrying capacities result:

$$R_{d,hold-down} = 1.1 \cdot \frac{60000}{1.3} = 50769.23 \text{ N}$$

$$R_{d,brackets/distributed\ nailing} = 1.1 \cdot \frac{12000}{1.3} = 10153.85 \text{ N}$$

Table 6.10 shows that the design load-carrying capacities of the tension and shear connections are actually calculated correctly.

Table 6.10 “More-Connections” file for line 11

New lines ID	New surfaces ID	Resistance connections property		
12	3	50769.23	50769.23	10153.85

6.5.2.3 “Standard” mode

When the user defines the connection properties with the “Standard” mode, the design load-carrying capacities of the hold-down and brackets or distributed nailing are easily calculated as:

$$R_{d,hold-down} = R_{k,single\ hold-down} \cdot \frac{k_{mod}}{\gamma_M} \cdot \text{number of hold-down}$$

$$R_{d,brackets} = R_{k,single\ bracket} \cdot \frac{k_{mod}}{\gamma_M} \cdot number\ of\ brackets$$

$$R_{d,distributed\ nailing} = R_{k,single\ nail} \cdot \frac{k_{mod}}{\gamma_M} \cdot number\ of\ nails$$

The characteristic load-carrying capacities of the single hold-down and single bracket or single nail are already set in the code as shown in Figures 6.9, 6.10 and 6.11.

Figure 6.15 shows the section of the code which calculates the design load-carrying capacities of the connections to write them in the “*More-Connections*” file.

```
set rd1 [expr $nchd1 * $Rk1 * $kmod / $gam]
set rd2 [expr $nchd2 * $Rk2 * $kmod / $gam]
set rd [expr $nc * $Rk * $kmod / $gam]
```

Figure 6.15 Design load-carrying capacities calculation for “Standard” mode

CHAPTER 7

MODELLING OF A COMPLEX STRUCTURE

This Chapter will be a presentation of the modelling of an X-Lam structure case study, starting from the preliminary design phase up to the actual modelling in GiD. The goal is indeed to show how the new problem type and the calculation work for a complex structure. Thus, the first sections focus on the preliminary design, necessary for the choice of panels and connections. Thereafter, the modelling in GiD will be presented by means of the new problem type, similarly to Chapter 5, but in a briefer manner, and mostly focusing on the engineering point of view. Finally, the results of the analysis are displayed.

7.1 Example introduction

The structure that will be presented in this Chapter consists of a two-storey building. It will be submitted to seismic action and analysed with an equivalent static analysis, even if it is not a regular building in neither plan nor elevation. Figures 7.1 and 7.2 show the front views and plans of the two-storey building that will be analysed.

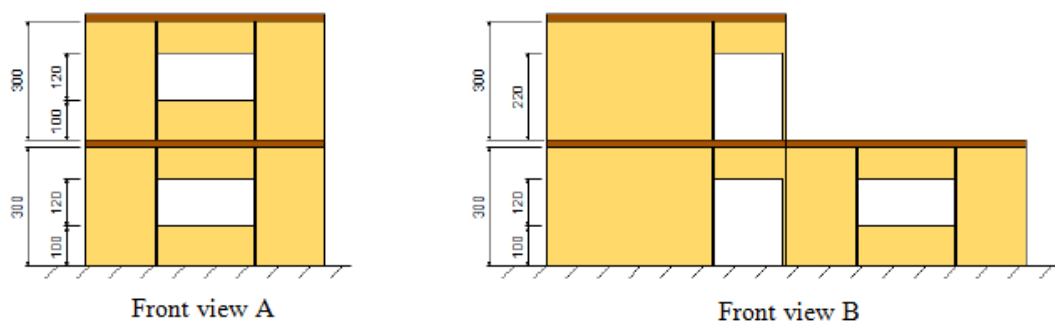


Figure 7.1(a) Building front views A and B

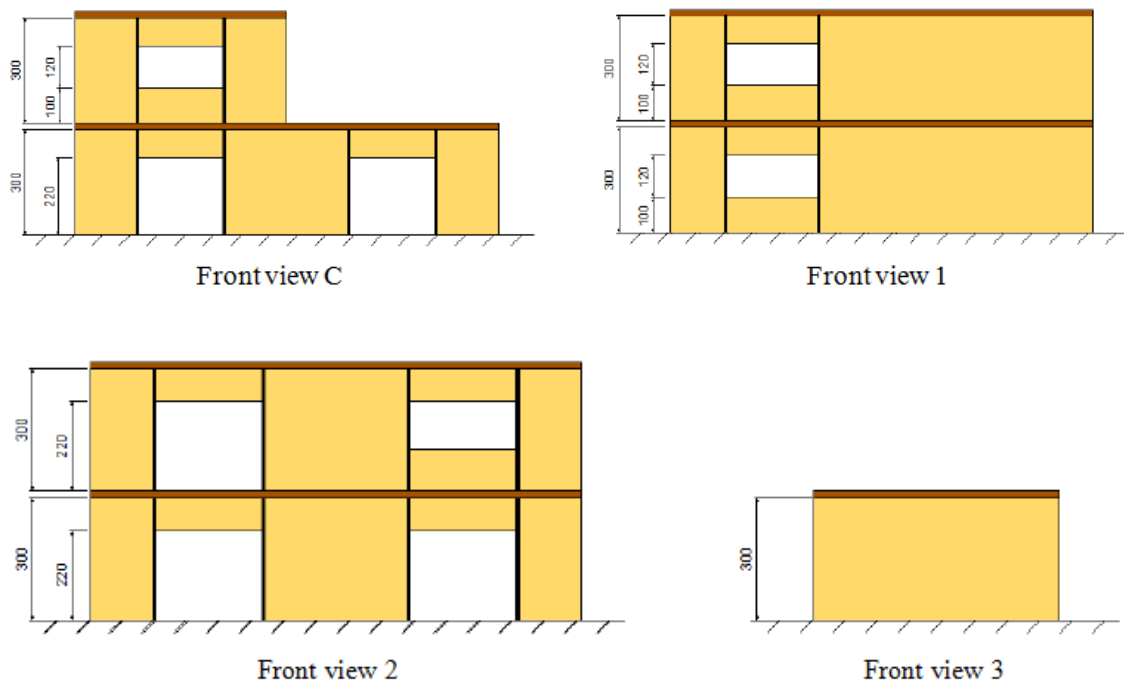


Figure 7.1(b) Building front views C, 1, 2 and 3

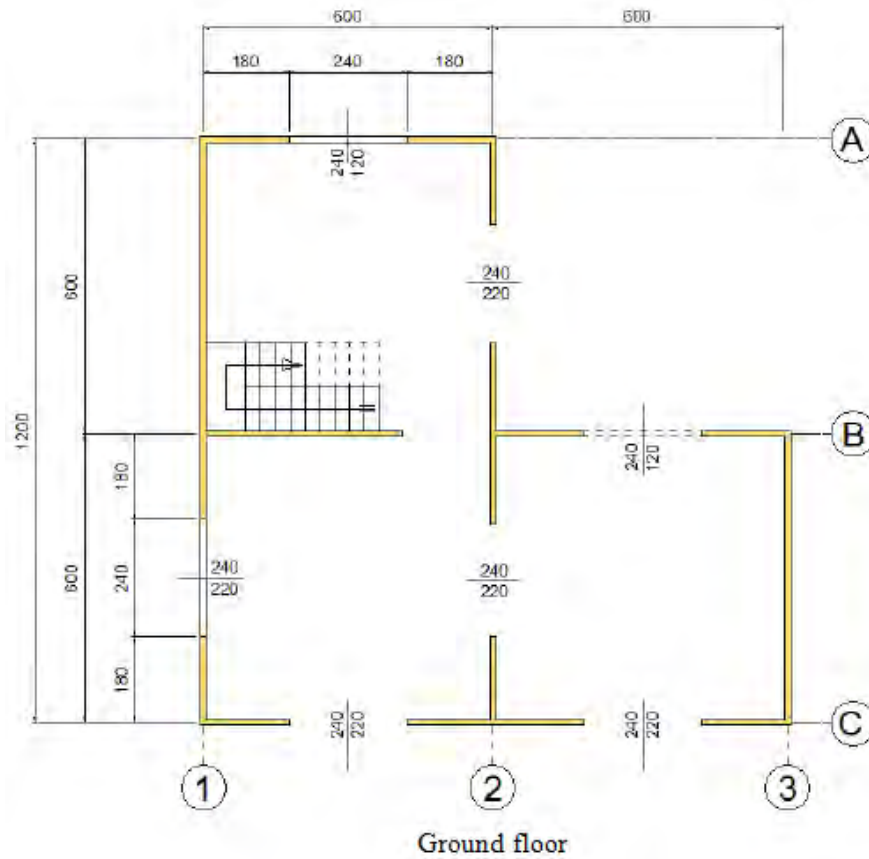


Figure 7.2(a) Ground floor plan

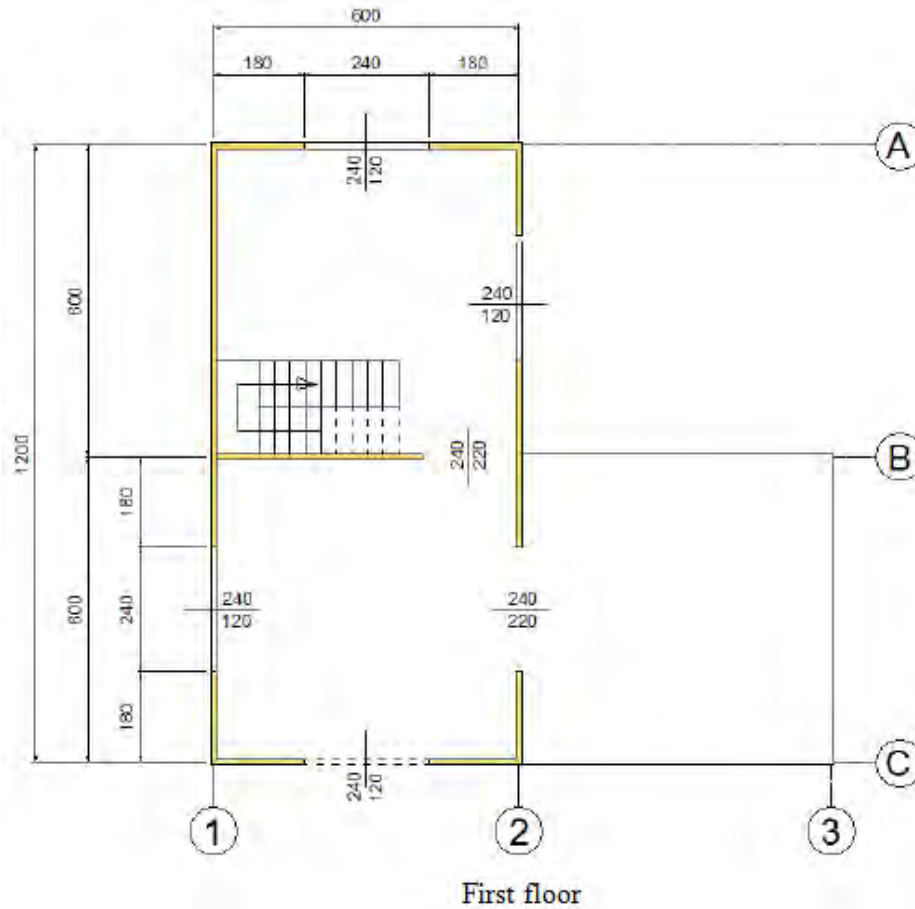


Figure 7.2(b) First floor plan

7.2 Preliminary design phase

The preliminary design consists on the static design of walls and slabs, considering the static loads, and their seismic design, by means of the calculation of the seismic weights, which, then, can be used for the equivalent static analysis. Moreover, it concerns the seismic design of the connections and the calculation of their resistance and stiffness.

7.2.1 Static design of X-Lam walls and slabs

The first step is the analysis of all the loads acting on the floor, on the roof and the load-bearing walls; they are summarised in Figure 7.3.

Floor		
g1: X-Lam panel (20 cm)	1,0 kN/mq	
g2: ceiling, floor, dividers	2,0 kN/mq	
q: overload residential building	2,0 kN/mq	($\Psi_2=0,3$)
TOT.	5,0 kN/mq	
$W_{seismic} = 1,0+2,0+0,3*2,0 = 3,6 \text{ kN/mq}$		
Roof		
g1: X-Lam panel (20 cm)	1,0 kN/mq	
g2: ceiling, insulating, roof	2,0 kN/mq	
q: overload snow (height<1000 m)	1,5 kN/mq	($\Psi_2=0$)
TOT.	4,5 kN/mq	
$W_{seismic} = 1,0+2,0+0*1,5 = 3,0 \text{ kN/mq}$		
Load-bearing walls		
g1: X-Lam panel (10 cm)	0,5 kN/mq	
g2: ceiling, insulating, covering	1,0 kN/mq	
TOT.	1,5 kN/mq	
$W_{seismic}: \text{distributed as floor load}$		

Figure 7.3 Loads analysis

The seismic loads lead from the combination required by D.M. 14-01-2008 "New Technical Standards for Construction" in Section 2.5.3, according to which they are calculated as:

$$E + G_1 + G_2 + P + \psi_{21}Q_{k1} + \psi_{22}Q_{k2}$$

where Q_{k1} represents the overhead for the slabs of residential use and Q_{k2} the overload on the roof; the component P is, in all cases, equal to zero; the seismic action E is calculated in Section 7.2.3. Without deepening too much about the coefficients $\psi_{2,j}$

provided by the code, for the snow overload $\psi_{2,j} = 0$, while for the overload of a residential building $\psi_{2,j} = 0.3$.

For the walls, the loads acting on a unit width are considered, shown in Figure 7.4. The total loads on the wall are indicated with a white arrow; they result:

$$g_k = 9.0 + 9.0 + 4.5 + 4.5 = 27 \text{ KN}$$

$$q_k = 4.5 + 6.0 = 10.5 \text{ KN}$$

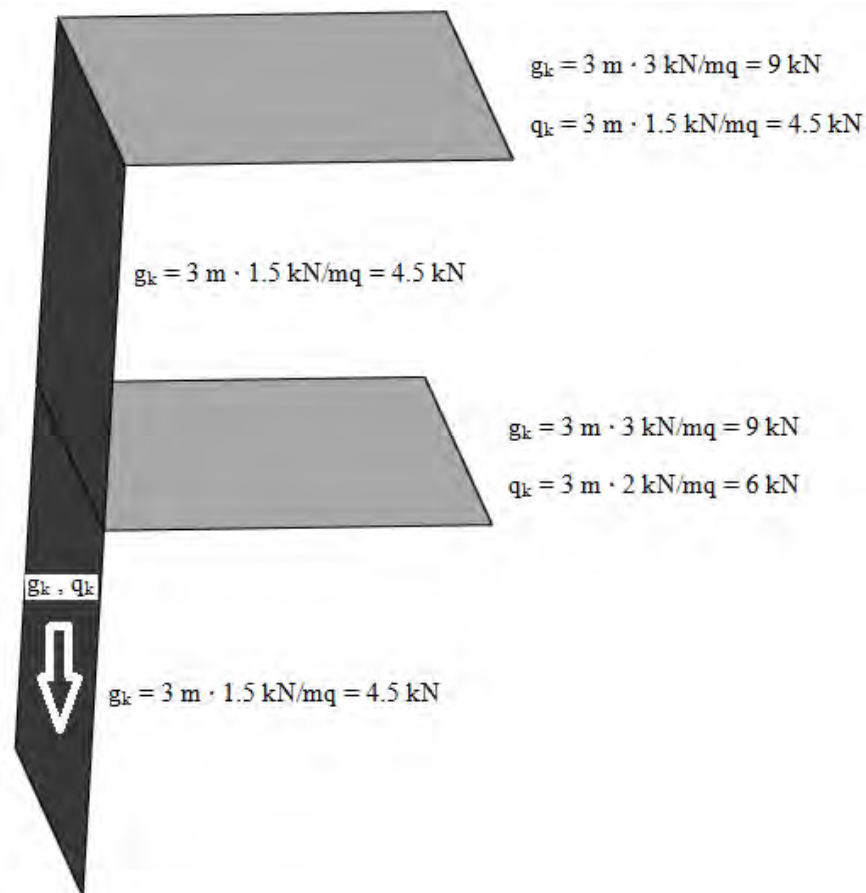


Figure 7.4 Loads representation on the geometry

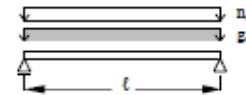
For the static preliminary design, the load charts provided by manufacturers (Rothoblaas) are used; these are only for preliminary design purposes and cannot substitute a structural analysis.

For the floor, known the permanent load ($g_k=2.0 \text{ kN/m}^2$) and the overload ($q_k=2.0 \text{ kN/m}^2$) and known the span length ($L=6\text{m}$), it results that 5 timber layers can be disposed, with a total thickness of the X-Lam floor equal to 196 mm (Figure 7.5).

For the roof, known the permanent load ($g_k=2.0 \text{ kN/m}^2$) and the overload ($q_k=1.5 \text{ kN/m}^2$) and known the span length ($L=6\text{m}$), it results that 5 timber layers can be disposed, with a total thickness of the X-Lam roof equal to 182 mm (Figure 7.5).

For the walls, considering a permanent load of 30 kN/m^2 , an overload of 10 kN/m^2 and an height of 2.9 m, it results, from the load charts provided by manufacturers (Rothoblaas), that 3 timber layers can be disposed, with a total thickness of the X-Lam wall equal to 97 mm (Figure 7.6).

Single-span floors



In accordance with approval Z 9.1-559
DIN 1052 (2008) and/or EN 1995-1-1 (2006)

Permanent load g_k [kN/m]	Imposed load q_k [kN/m]	SPAN OF SINGLE-SPAN BEAM l								
		3.00 m	3.50 m	4.00 m	4.50 m	5.00 m	5.50 m	6.00 m	6.50 m	7.00 m
1.00	1.00	83 L3s	83 L3s	103 L3s	103 L3s	112 L3s	150 L5s	165 L5s	182 L5s	211 L5s
	2.00	74 L3s	97 L3s	97 L3s	126 L3s	126 L3s		196 L5s	196 L5s	209 L7s-2
	2.80	83 L3s	97 L3s	103 L3s	119 L3s	138 L5s	182 L5s	211 L5s	223 L7s-2	
	3.50			112 L3s	126 L3s	150 L5s				249 L7s-2
	4.00	97 L3s	103 L3s	119 L3s	150 L5s	150 L5s	165 L5s	196 L5s	209 L7s-2	209 L7s-2
5.00	74 L3s	97 L3s	103 L3s	112 L3s	126 L3s	150 L5s	182 L5s	209 L7s-2	223 L7s-2	
1.50	1.00	83 L3s	83 L3s	103 L3s	112 L3s	126 L3s	150 L5s	182 L5s	196 L5s	211 L5s
	2.00	74 L3s	97 L3s	97 L3s	119 L3s	126 L3s		196 L5s	196 L5s	209 L7s-2
	2.80	83 L3s	97 L3s	103 L3s	119 L3s	138 L5s	165 L5s	196 L5s	209 L7s-2	
	3.50			112 L3s	126 L3s	150 L5s				223 L7s-2
	4.00	97 L3s	103 L3s	119 L3s	150 L5s	150 L5s	182 L5s	211 L5s	211 L5s	223 L7s-2
5.00	74 L3s	97 L3s	103 L3s	112 L3s	126 L3s	150 L5s	182 L5s	209 L7s-2	223 L7s-2	
2.00	1.00	83 L3s	83 L3s	103 L3s	112 L3s	126 L3s	150 L5s	182 L5s	196 L5s	211 L5s
	2.00	74 L3s	97 L3s	97 L3s	119 L3s	126 L3s		196 L5s	196 L5s	209 L7s-2
	2.80	83 L3s	97 L3s	103 L3s	119 L3s	138 L5s	165 L5s	196 L5s	209 L7s-2	
	3.50			112 L3s	126 L3s	150 L5s				223 L7s-2
	4.00	97 L3s	103 L3s	119 L3s	150 L5s	150 L5s	182 L5s	211 L5s	211 L5s	223 L7s-2
5.00	74 L3s	97 L3s	103 L3s	112 L3s	126 L3s	150 L5s	182 L5s	209 L7s-2	223 L7s-2	

Figure 7.5 Preliminary design table for single-span floors (Rothoblaas)

External walls



In accordance with approval Z 9-1-559
DIN 1052 (2008) and/or EN 1995-1-1 (2005)

Permanent load [KN/m]	Imposed load [KN/m]	HEIGHT (buckle length l)											
		2.46 m				2.76 m				2.86 m			
g_k	m_k	R 0	R 30	R 60	R 90	R 0	R 30	R 60	R 90	R 0	R 30	R 60	R 90
20.00	10.00		92 C3s	85 C3s	97 C3s			97 C3s	97 C3s			97 C3s	97 C3s
	20.00					67 C3s				67 C3s			
	30.00	67 C3s		97 C3s									
	40.00		97 C3s	96 C3s	138 C3s			96 C3s	138 C3s		97 C3s	96 C3s	138 C3s
	50.00	88 C3s				88 C3s				88 C3s			
30.00	10.00			97 C3s	97 C3s			97 C3s		67 C3s		97 C3s	
	20.00												
	30.00	67 C3s											
	40.00		97 C3s	96 C3s	138 C3s			96 C3s	138 C3s		97 C3s	96 C3s	138 C3s
	50.00	88 C3s				88 C3s				88 C3s			
40.00	10.00							97 C3s		67 C3s			
	20.00												
	30.00	67 C3s											
	40.00		97 C3s	96 C3s	138 C3s			96 C3s	138 C3s		97 C3s	96 C3s	138 C3s
	50.00	88 C3s				88 C3s				88 C3s			
60.00											138 C3s		

Figure 7.6 Preliminary design table for external walls (Rothoblaas)

7.2.2 Seismic design of X-Lam walls and slabs

In order to calculate the seismic weights, the Area and Perimeter of the ground and first floor are first determined, resulting:

- ground floor: $A = 108 \text{ m}^2$, $P_{\text{walls}} = 60 \text{ m}$;
- first floor: $A = 72 \text{ m}^2$, $P_{\text{walls}} = 42 \text{ m}$.

Then, the seismic weights afferent to the floor and the roof, depending on their afferent heights (Figure 7.7), can be calculated as:

$$W = \text{Area} \cdot \text{seismic load} + \text{Perimeter} \cdot \text{afferent height} \cdot \text{total walls load}$$

In the case of the floor, it results:

$$W_F = 108 \cdot 3.6 + 60 \cdot 3 \cdot 1.5 = 658.8 \text{ kN}$$

$$w_F = W_F/Area = 658.8/108 = 6.1 \text{ kN/m}^2$$

In the case of the roof, it results:

$$W_R = 72 \cdot 3.0 + 42 \cdot 1.5 \cdot 1.5 = 310.5 \text{ kN}$$

$$w_R = W_R/Area = 310.5/72 = 4.3 \text{ kN/m}^2$$

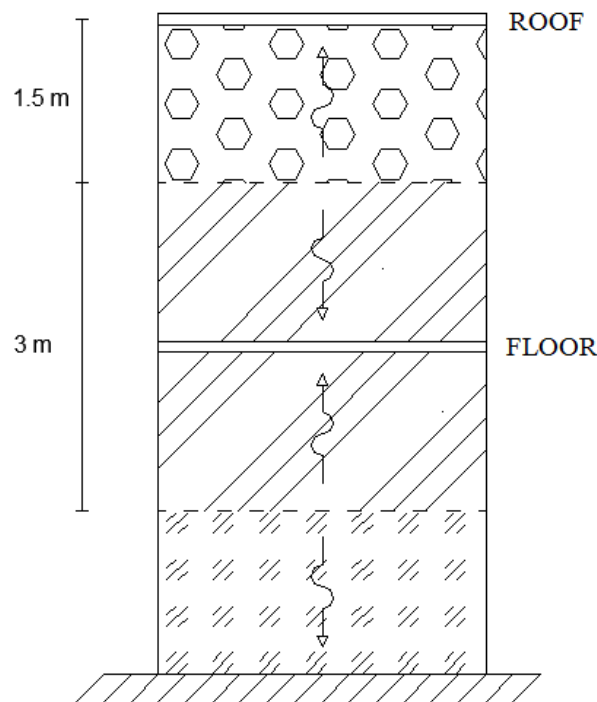


Figure 7.7 *Afferent heights of the slabs*

Once the seismic weights are known, an equivalent static analysis can be used to evaluate the storeys seismic forces. It could not be used because the building is not regular in plan and elevation; nonetheless, it is used anyway, with the sole purpose of testing the capabilities and reliability of the software.

7.2.3 Equivalent static analysis

Being the response history and response spectrum analyses still not available in Kratos solver and non-linear analyses not available in the software because of the use of linear elastic spring elements, an equivalent static analysis is performed.

According to EN 1998-1: 2004 (8.3), depending on their ductile behaviour and energy dissipation capacity under seismic actions, timber buildings shall be assigned to one of the three ductility classes L, M or H, as given in the Table 7.1.

Table 7.1 Design concept, structural types and upper limit values of the behaviour factors for the three ductility classes (EN 1998-1: 2004)

Design concept and ductility class	q	Examples of structures
Low capacity to dissipate energy - DCL	1,5	Cantilevers; Beams; Arches with two or three pinned joints; Trusses joined with connectors.
Medium capacity to dissipate energy - DCM	2	Glued wall panels with glued diaphragms, connected with nails and bolts; Trusses with doweled and bolted joints; Mixed structures consisting of timber framing (resisting the horizontal forces) and non-load bearing infill.
	2,5	Hyperstatic portal frames with doweled and bolted joints (see 8.1.3(3)P).
High capacity to dissipate energy - DCH	3	Nailed wall panels with glued diaphragms, connected with nails and bolts; Trusses with nailed joints.
	4	Hyperstatic portal frames with doweled and bolted joints (see 8.1.3(3)P).
	5	Nailed wall panels with nailed diaphragms, connected with nails and bolts.

Being the building non-regular in elevation the q -values listed in Table 7.1 should be reduced by 20%, but need not be taken less than $q=1,5$. For the analysed building, it results $q=1.6$.

The two-storey building is located in Geron, Friuli Venezia Giulia, Italy, where the ground type is B, and the elastic response spectrum for the ultimate limit state (ULS) and damage limit state (DLS) is the one shown in Figure 7.8.

According to EN 1998-1:2004 (4.3.3.2.2), being a building with height less than 40 m, the value of the fundamental period of vibration T_1 may be approximated by the following expression:

$$T_1 = C_t \cdot H^{3/4} = 0.050 \cdot 6^{3/4} = 0.192 \text{ s}$$

where

C_t is 0,085 for moment resistant space steel frames, 0,075 for moment resistant space concrete frames and eccentrically braced steel frames and 0,050 for all other structures (the last is the case of X-Lam buildings);

H is the height of the building, in m, from the foundation or from the top of a rigid basement.

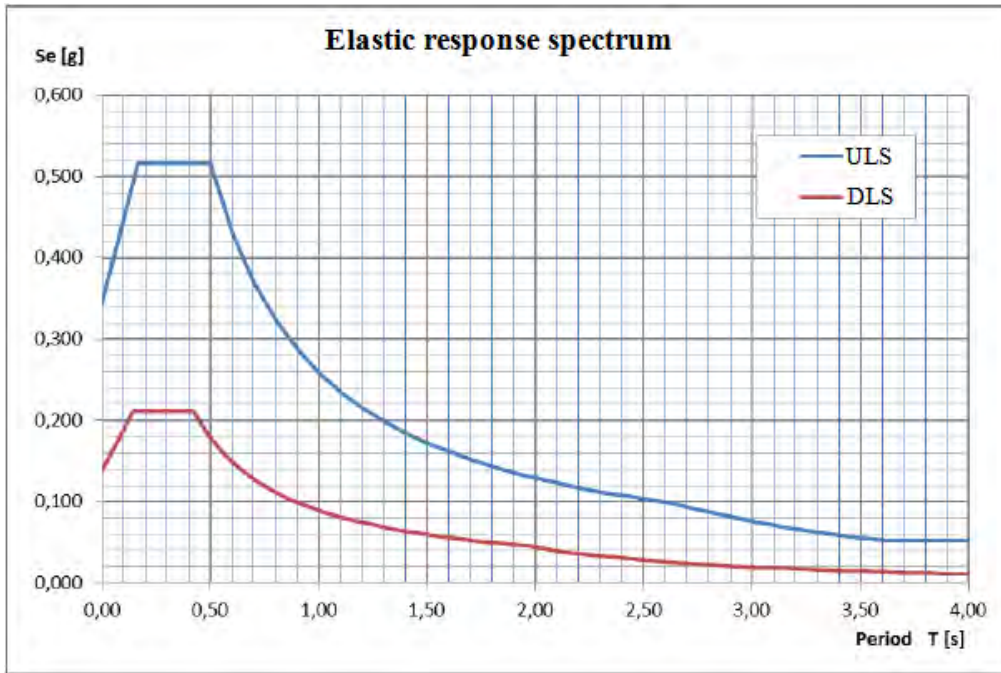


Figure 7.8 Elastic response spectrum for Gerona, Friuli Venezia Giulia, Italy

The seismic base shear force F_b , for each horizontal direction in which the building is analysed, shall be determined using the following expression:

$$F_b = S_d(T_1) \cdot m \cdot \lambda = 0.45g \cdot \frac{W_{TOT}}{g} \cdot 0.85$$

where

$S_d(T_1)$ is the ordinate of the design spectrum;

T_1 is the fundamental period of vibration of the building for lateral motion in the direction considered;

m is the total mass of the building, above the foundation or above the top of a rigid basement;

λ is the correction factor, whose value is equal to 0,85 if $T_1 < 2 T_C$ and the building has more than two storeys, otherwise $\lambda = 1,0$.

The total seismic load can be determined as the sum of the roof seismic load and the floor one, just calculated before:

$$W_{TOT} = W_{floor} + W_{roof} = 658.8 + 310.5 = 969.3 \text{ kN}$$

In this way, the seismic base shear force F_b results:

$$F_b = S_d(T_1) \cdot m \cdot \lambda = 0.45g \cdot \frac{W_{TOT}}{g} \cdot 0.85 = 0.45 \cdot 969.3 \cdot 0.85 = 371 \text{ kN}$$

The seismic action effects can be determined by applying horizontal forces F_i to all storeys:

$$F_i = F_b \cdot \frac{s_i \cdot m_i}{\sum s_j \cdot m_j}$$

where

F_i is the horizontal force acting on storey i ;

F_b is the seismic base shear;

s_i, s_j are the displacements of the storey masses m_i, m_j in the fundamental mode shape.

When the fundamental mode shape is approximated by horizontal displacements increasing linearly along the height, the horizontal forces F_i can be taken as being given by:

$$F_i = F_b \cdot \frac{z_i \cdot W_i}{\sum z_j \cdot W_j}$$

where z_i, z_j are the heights of the masses m_i and m_j above the level of application of the seismic action (foundation or top of a rigid basement).

Table 7.2 Storeys seismic forces

	Wi (kN)	zi (m)	Wi · zi (kNm)	Fi (kN)	Fi * 1,25 (kN)
Roof	310,5	6	1863,0	180	225
Floor	658,8	3	1976,4	191	239
TOT.	969,3		3839,4	371	

For the preliminary design of the connections, the storeys seismic forces (Table 7.2) are multiplied by an approximate factor that takes into account the non-regularity of the structure:

$$W_R = 180 \cdot 1.25 = 225 \text{ kN}$$

$$W_F = 191 \cdot 1.25 = 239 \text{ kN}$$

The forces should be applied at the barycentre of the storeys: this means to apply the force of the roof at the point with coordinates (3, 6, 6) and the force of the floor at the point (5, 5, 3), with reference to Figure 7.9.

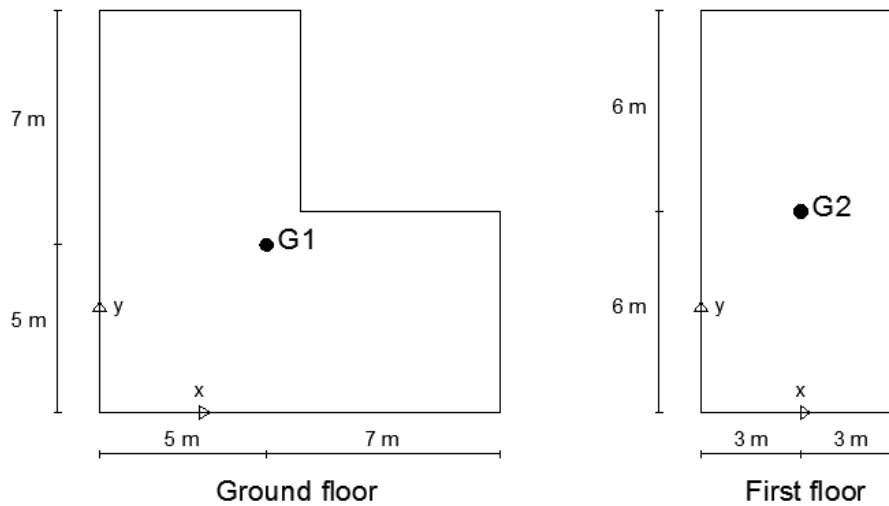


Figure 7.9 Barycentre position at the two storeys of the building

The seism can act in any direction, in particular not necessarily in x or y-direction: to take into account this effect, two different analyses are considered, one with the seismic forces applied in x-direction and the 30% of the same forces applied in y-direction, another the other way around.

7.2.4 Connections seismic design

The distribution of the seismic forces on the walls does not depend on the X-Lam panel characteristics (thickness, inertia, length), but it only depends on the connections at the base and at the interstory.

7.2.4.1 Shear connections

The calculation of the resistance of the shear connections can be made preliminarily by means of the manufacturers catalogues. Brackets type WBR100 with total nailing and 12 nails Anker $\Phi 4 \times 60$, referring to the technical data sheet Rothoblaas (Figure 7.10), will be used.

GIUNZIONE LEGNO - CEMENTO

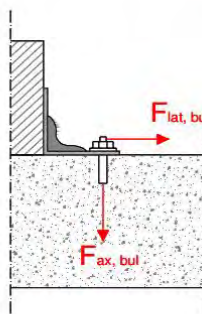
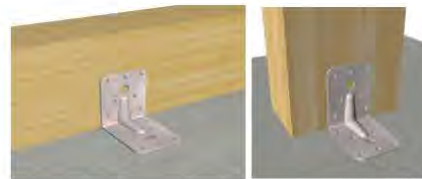
Fissaggio Cemento - Trave o Cemento - Pilastro

CONNETTORI LATO LEGNO:

CHIODI ANKER
 $\Phi 4,0 \times 60$



VITI SPECIALI
 $\Phi 5,0 \times 60$



$F_{lat, bul}$ Forza di taglio agente sull'ancorante
 $F_{ax, bul}$ Forza assiale sull'ancorante

Il collegamento dell'ancoraggio al cemento è da verificare a parte sulla base delle forze sollecitanti l'ancorante stesso



● = Chiodatura parziale
●+● = Chiodatura totale

Resistenza R_{20} - 1 angolare per giunzione

			$F_{20, max}^{**}$	
	Chiod. tot.	$R_{20A, TOT}$	kN	8,94
	Chiod. parz.	$R_{20A, PARZ}$	kN	6,07
	Viti tot.	$R_{20B, TOT}$	kN	11,72
	Viti parz.	$R_{20B, PARZ}$	kN	7,81

Figure 7.10 Selected shear connection (Rothoblaas)

According to EN.1995.1.1.2009 (2.4.3), the design value of the load-carrying capacity can be obtained as:

$$R_d = \frac{R_k \cdot k_{mod}}{\gamma_M} = \frac{8.9 \cdot 1.1}{1.3} = 7.56 \text{ kN}$$

The number of brackets needed at the two floors is:

- at the ground floor:

$$n = \frac{W}{R_{d,ang}} = \frac{W_R + W_F}{R_{d,ang}} = \frac{225 + 239}{7.56} \approx 62 \text{ brackets}$$

- at the interstory:

$$n = \frac{W_R}{R_{d,ang}} = \frac{225}{7.56} \approx 30 \text{ brackets}$$

The distribution of the brackets is made depending on the length of the seismic resistant walls:

$$L_{\text{seismic resistant walls_ground floor_x}} = 18.6 \text{ m} \rightarrow 62/18.6 = 3.3 \text{ angle brackets/m}$$

$$L_{\text{seismic resistant walls_ground floor_y}} = 22.8 \text{ m} \rightarrow 62/22.8 = 2.7 \text{ angle brackets/m}$$

$$L_{\text{seismic resistant walls_first floor_x}} = 11.4 \text{ m} \rightarrow 30/11.4 = 2.6 \text{ angle brackets/m}$$

$$L_{\text{seismic resistant walls_first floor_y}} = 16.8 \text{ m} \rightarrow 30/16.8 = 1.8 \text{ angle brackets/m}$$

7.2.4.2 Tension connections

The calculation of the resistance of the tension connections can be made preliminarily by means of the manufacturers catalogues. The following types of hold-down, referring to the technical data sheet Rothoblaas (Figure 7.11), will be used:

- hold-down WHT 340 with total nailing and 20 nails Anker $\Phi 4 \times 60$;
- hold-down WHT 440 with total nailing and 30 nails Anker $\Phi 4 \times 60$;
- hold-down WHT 540 with total nailing and 42 nails Anker $\Phi 4 \times 60$;

- hold-down WHT 620 with total nailing and 52 nails Anker $\Phi 4 \times 60$.

Similarly to the shear connections and according to EN.1995.1.1.2009 (Art. 2.4.3), the design value of the load-carrying capacity can be obtained as:

$$R_d = \frac{R_k \cdot k_{mod}}{\gamma_M}$$

$$R_{d,WHT340} = \frac{R_{k,WHT440} \cdot k_{mod}}{\gamma_M} = \frac{38.6 \cdot 1.1}{1.3} = 33 \text{ kN}$$

$$R_{d,WHT440} = \frac{R_{k,WHT440} \cdot k_{mod}}{\gamma_M} = \frac{57.9 \cdot 1.1}{1.3} = 49 \text{ kN}$$

$$R_{d,WHT540} = \frac{R_{k,WHT440} \cdot k_{mod}}{\gamma_M} = \frac{81.1 \cdot 1.1}{1.3} = 69 \text{ kN}$$

$$R_{d,WHT620} = \frac{R_{k,WHT620} \cdot k_{mod}}{\gamma_M} = \frac{100.4 \cdot 1.1}{1.3} = 85 \text{ kN}$$

WHT - CHIODATURA PARZIALE			Resistenza caratteristica a trazione							
TYP WHT	Fissaggio Fori Ø 5 (connettori)		R _k lato legno		R _k lato acciaio					
	Chiodi Anker	Viti Speciali	n _{conn} [pz.]	R _k legno [kN]	Rondella	R _k acciaio [kN]				
340	Ø 4,0 x 40	Ø 5,0 x 40	14	22,0	-	42,0				
	Ø 4,0 x 60	Ø 5,0 x 50		27,0						
440	Ø 4,0 x 40	Ø 5,0 x 40	20	31,4	-	42,0				
	Ø 4,0 x 60	Ø 5,0 x 50		38,6						
540	Ø 4,0 x 40	Ø 5,0 x 40	26	40,8	-	42,0				
	Ø 4,0 x 60	Ø 5,0 x 50		50,2						
620	Ø 4,0 x 40	Ø 5,0 x 40	32	50,2	-	42,0				
	Ø 4,0 x 60	Ø 5,0 x 50		61,8						

Utilizzando 2 angolari TYP WHT per singola giunzione, le resistenze di progetto raddoppiano.

WHT - CHIODATURA TOTALE			Resistenza caratteristica a trazione							
TYP WHT	Fissaggio Fori Ø 5 (connettori)		R _k lato legno		R _k lato acciaio					
	Chiodi Anker	Viti Speciali	n _{conn} [pz.]	R _k legno [kN]	Rondella	R _k acciaio [kN]				
340	Ø 4,0 x 40	Ø 5,0 x 40	20	31,4	-	42,0				
	Ø 4,0 x 60	Ø 5,0 x 50		38,6						
440	Ø 4,0 x 40	Ø 5,0 x 40	30	47,1	* H	63,4				
	Ø 4,0 x 60	Ø 5,0 x 50		57,9	10 mm					
540	Ø 4,0 x 40	Ø 5,0 x 40	42	65,9	* H	63,4				
	Ø 4,0 x 60	Ø 5,0 x 50		81,1	10 mm					
620	Ø 4,0 x 40	Ø 5,0 x 40	52	81,6	** H	85,2				
	Ø 4,0 x 60	Ø 5,0 x 50		100,4	20 mm					

* Rondella ULS505610 ** Rondella ULS707720

Figure 7.11 Selected tension connections (Rothoblaas)

The stresses on the hold-down can be calculated with equilibrium relationships: the rigid rotation of the first floor and the rocking effect on one single wall should be considered.

Concerning the rigid rotation of the first floor, the overturning and stabilising moment result:

$$M_{overt} = W_{_R} \cdot h = 225 \cdot 3 = 675 \text{ kNm}$$

$$M_{stab} = W_{roof} \cdot \frac{B}{2} = 310.5 \cdot \frac{6}{2} = 931.5 \text{ kNm}$$

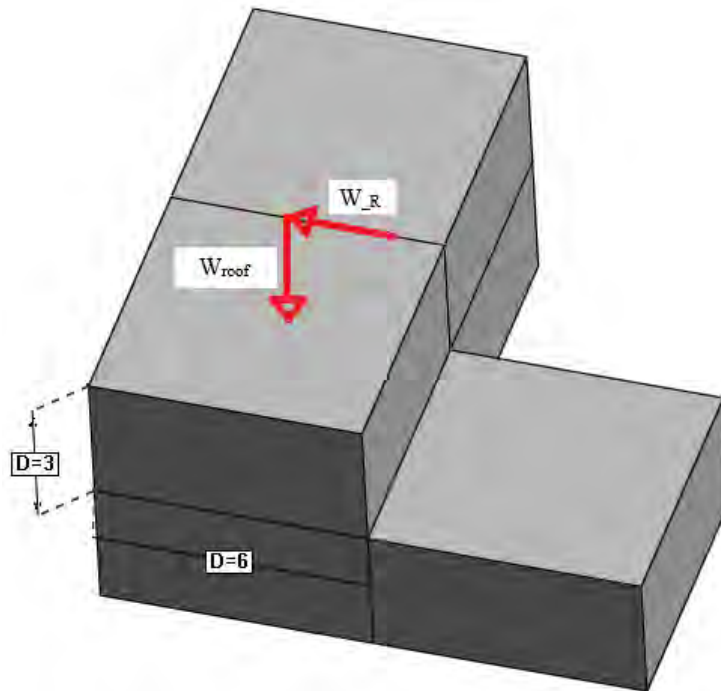


Figure 7.12 Forces causing the first floor rigid rotation

Since the stabilising effect of the self-weight is greater than the overturning one caused by the seism (Figure 7.12), there is not rigid overturning of parts of the building.

Concerning the rocking effect, for each wall the resulting forces due to the rocking effect and the stresses can be calculated; depending on these, which type of hold-down disposing on that wall can be decided. All these quantities are summarized in Table 7.3, with reference to Figure 7.13.

Table 7.3(a) Summary of forces, stresses and hold-down disposed on walls in x-direction

DIRECTION X									
	Wall 1	Wall 2	Wall 3	Wall 4	Wall 5	Wall 6	Wall 7	Wall 8	Wall 9
Length (m)	1,8	1,8	4,2	1,8	1,8	1,8	1,8	1,8	1,8
Force roof (kN)	37,5	37,5	75,0	37,5	37,5	0,0	0,0	0,0	0,0
Force floor (kN)	23,9	23,9	38,2	23,9	23,9	28,6	28,6	23,9	23,9
V first storey (kN)	37,5	37,5	75,0	37,5	37,5	0,0	0,0	0,0	0,0
V base (kN)	61,3	61,3	113,1	61,3	61,3	28,6	28,6	23,9	23,9
M first storey (kN)	112,4	112,4	224,9	112,4	112,4	0,0	0,0	0,0	0,0
M base (kN)	296,4	296,4	564,3	296,4	296,4	85,9	85,9	71,6	71,6
N first storey (kN)	15,6	15,6	58,9	15,6	15,6	0,0	0,0	0,0	0,0
N base (kN)	33,9	33,9	128,7	33,9	33,9	18,3	18,3	18,3	18,3
Rhd first storey (kN)	54,7	54,7	24,1	54,7	54,7	0,0	0,0	0,0	0,0
Rhd base (kN)	147,7	147,7	70,0	147,7	147,7	38,6	38,6	30,6	30,6
HD first storey	1*540	1*540	1*340	1*540	1*540	NO	NO	NO	NO
HD base	2*620	2*620	1*620	2*620	2*620	1*440	1*440	1*340	1*340

Table 7.3(b) Summary of forces, stresses and hold-down disposed on walls in y-direction

DIRECTION Y						
	Wall 1	Wall 2	Wall 3	Wall 4	Wall 5	Wall 6
Length (m)	7,8	1,8	1,8	3,6	1,8	6,0
Force roof (kN)	91,4	21,1	28,1	56,2	28,1	0,0
Force floor (kN)	77,5	17,9	23,9	47,7	23,9	47,7
V first storey (kN)	91,4	21,1	28,1	56,2	28,1	0,0
V base (kN)	168,9	39,0	52,0	103,9	52,0	47,7
M first storey (kN)	274,1	63,2	84,3	168,7	84,3	0,0
M base (kN)	780,7	180,2	240,2	480,5	240,2	143,1
N first storey (kN)	49,7	15,6	15,6	31,2	15,6	0,0
N base (kN)	107,5	33,9	33,9	84,1	50,1	39,5
Rhd first storey (kN)	10,3	27,3	39,0	31,2	39,0	0,0
Rhd base (kN)	46,4	83,1	116,5	91,4	108,4	4,1
HD first storey	1*340	1*340	1*440	1*340	1*340	NO
HD base	1*440	1*620	2*540	1*620	1*540	1*340

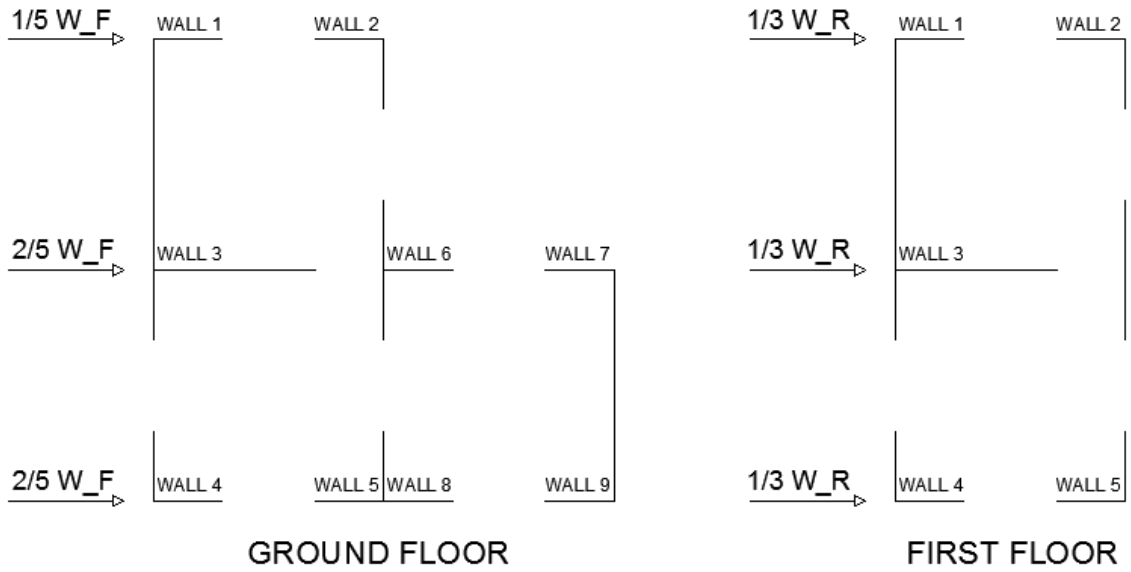


Figure 7.13(a) Scheme of walls and relative forces in x-direction

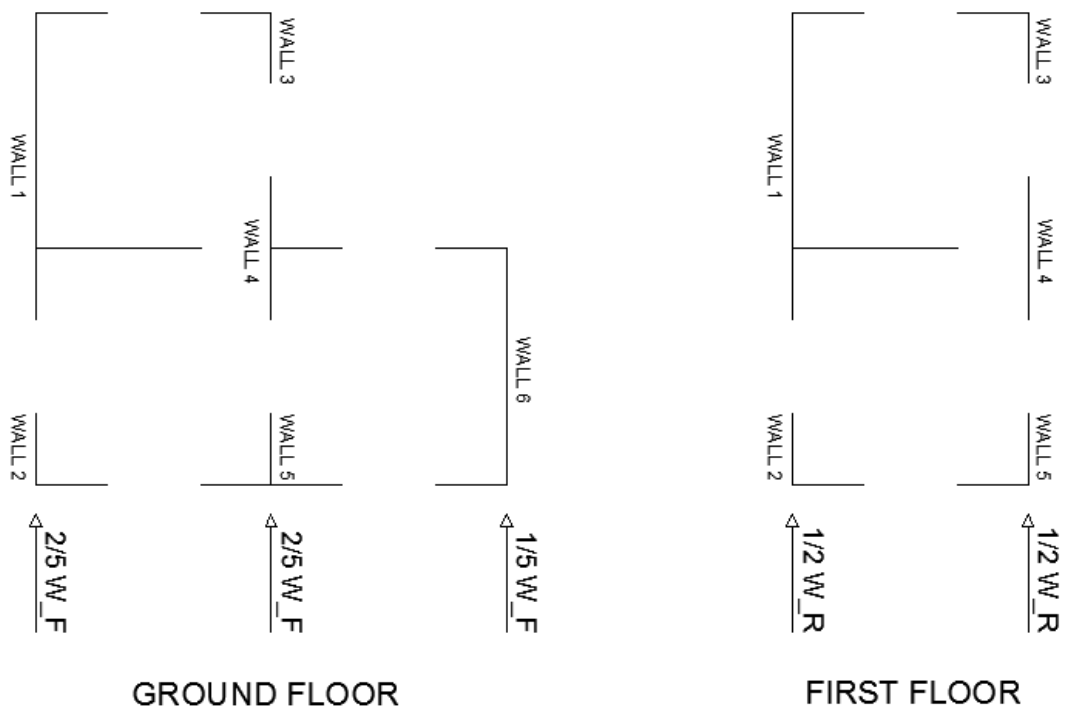


Figure 7.13(b) Scheme of walls and relative forces in y-direction

7.2.4.3 Connections stiffness

Modelling is very important to properly evaluate the connections stiffness because it affects in a significant way the global response of the building and the distribution of seismic forces.

Assuming to use a C22 timber ($\rho_m = 410 \text{ kg/m}^3$), the service stiffness per shear plane per fastener, for timber-to-steel connections, can be calculated as:

$$k_{ser} = 2 \cdot \frac{\rho_m^{1.5} \cdot d^{0.8}}{30} = 2 \cdot \frac{410^{1.5} \cdot 4^{0.8}}{30} = 1.7 \cdot 10^6 \text{ N/m}$$

Multiplying the service stiffness by the number of nails of the tension and shear connections, the stiffness of a single connection can be calculated. It is important to stress the point that the modelled stiffness of the hold-down is equal to two times the real one, taking into account the difference in behaviour, under the action of horizontal forces, of a single modelled panel compared with the same in a real situation (Section 4.3).

7.3 Modelling

This Section will concern on the modelling of the structure in GiD, especially focusing on the engineering perspective since the features of the problem type have been described in detail in Chapter 5.

7.3.1 Geometry

The geometry can be drawn in GiD using the plans and front views of Section 7.1; it is shown in Figure 7.14 from different points of view.

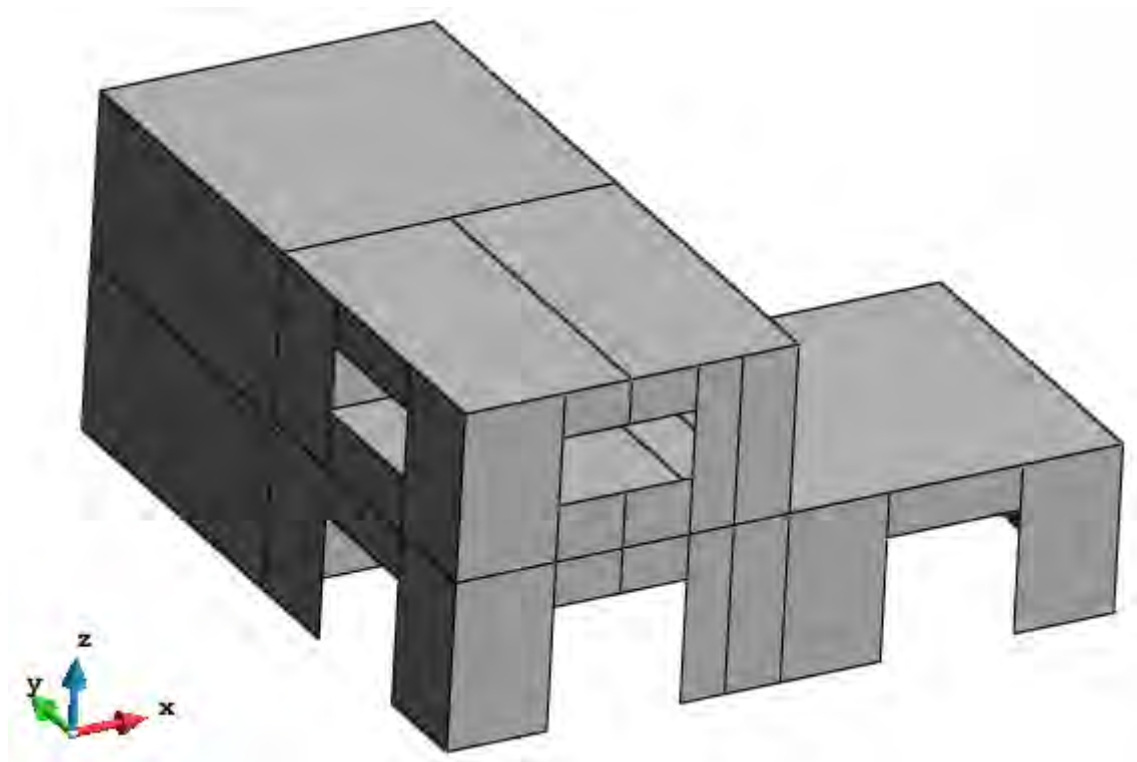


Figure 7.14(a) *Building geometry in GiD (a)*

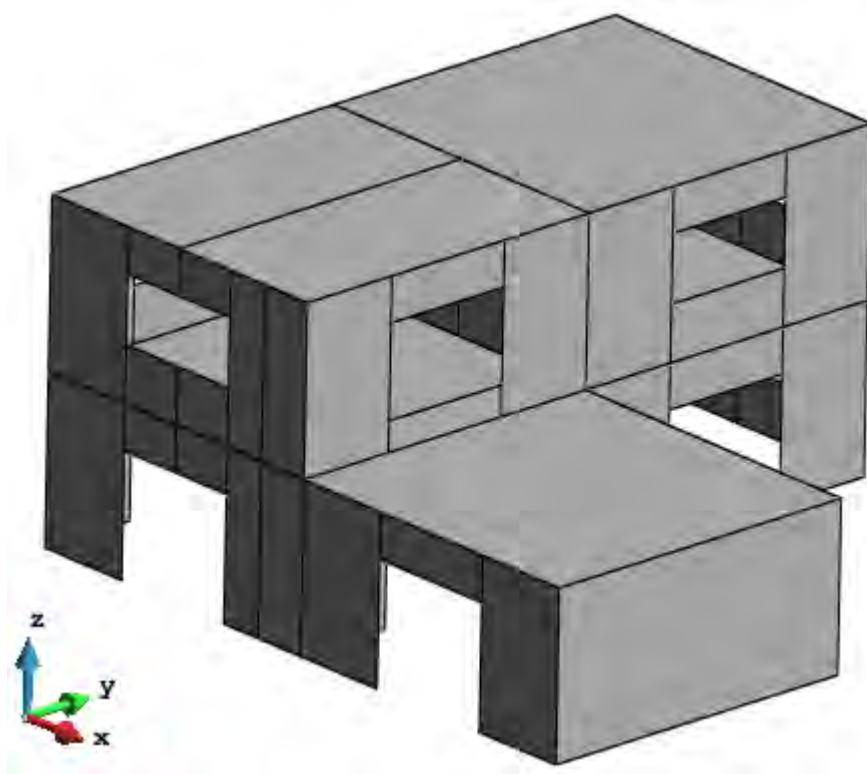


Figure 7.14(b) *Building geometry in GiD (b)*

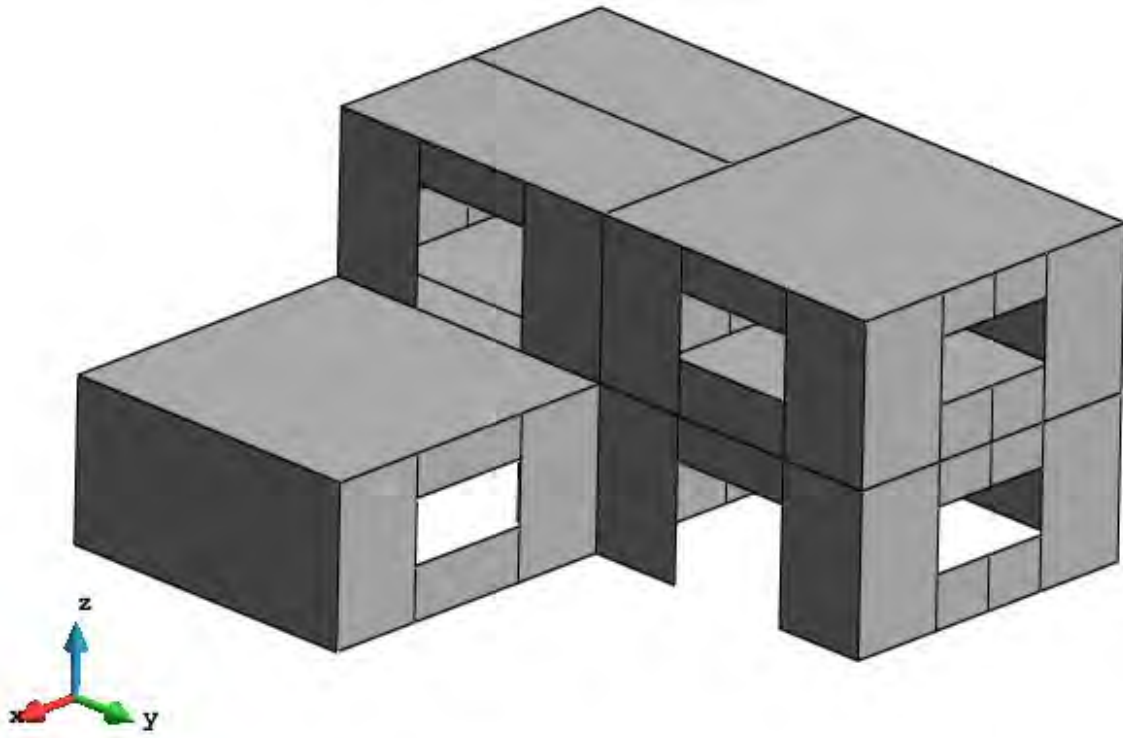


Figure 7.14(c) *Building geometry in GiD (c)*

Figure 7.14 shows that the slabs are drawn with different surfaces; this is to guarantee that the barycentre points of the first storey and the roof belong to the surfaces, hence the seismic forces can be applied on them. To identify these points, starting in drawing from the bottom, also some walls are drawn with different surfaces.

7.3.2 Material and elements properties

Relatively to the material, timber C22 is used, this is already defined in GiD material database, with all its properties already set.

Relatively to the elements properties, two different shell properties for the slabs and walls (Figure 7.15) are defined, since they have a different thickness, meaning also two solid properties. Additionally, the slabs are assumed to have the grain orientation of the outer layer in x-direction (angle = 0°), while for the walls the grain orientation of the outer layer is assumed in z-direction (angle = 90°).

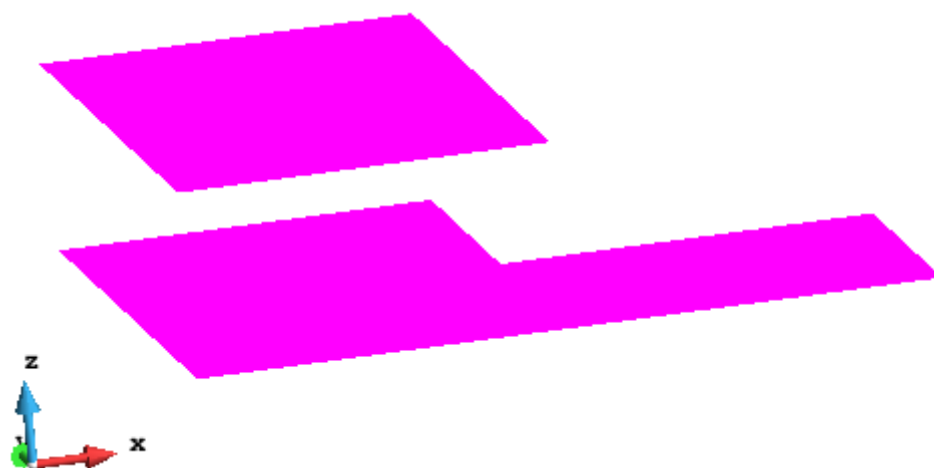


Figure 7.15(a) *Slabs shell elements*

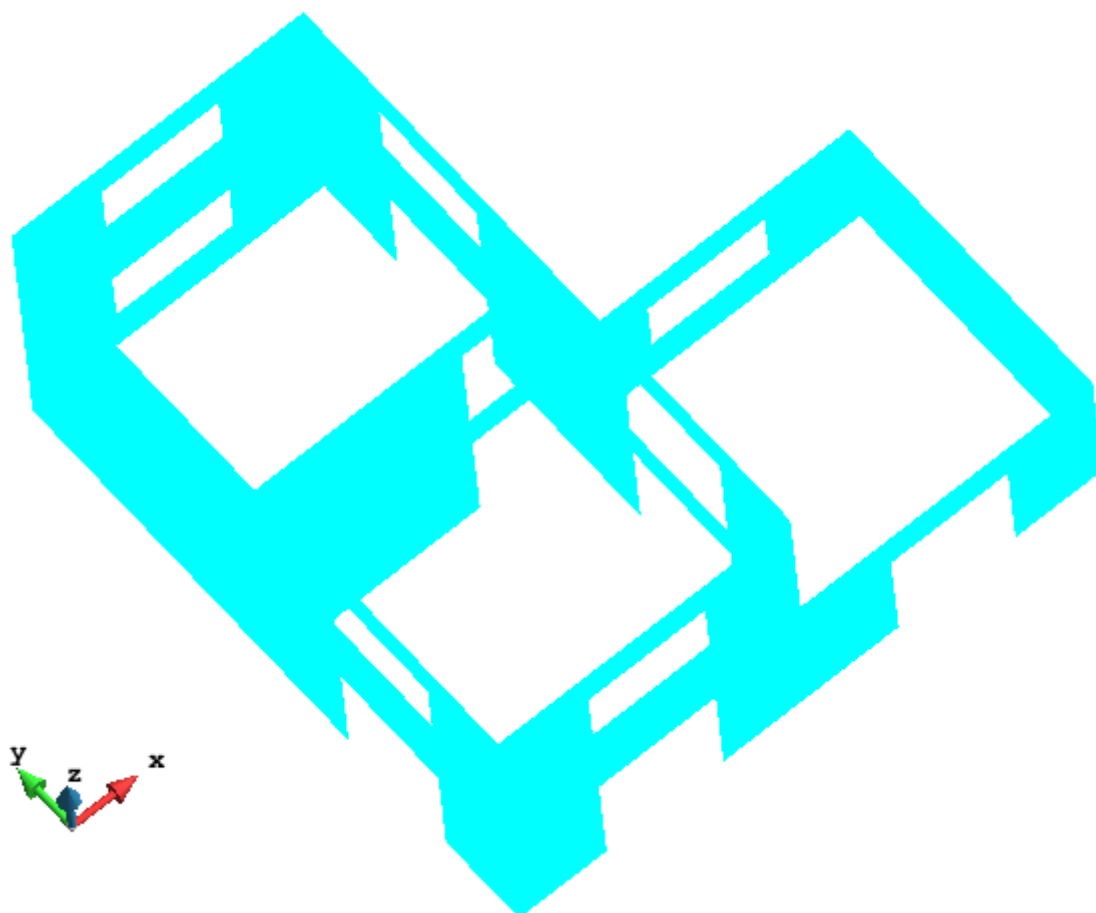


Figure 7.15(b) *Walls shell elements*

Moreover, two different beam properties are defined to distinguish between the curbs, which have a greater section of 0.15m·0.15m, from the ordinary ones which have a small section of 0.001m·0.001m (Figure 7.16).

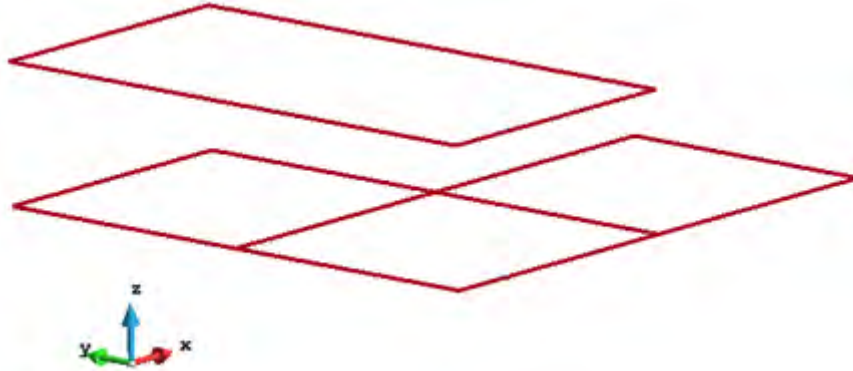


Figure 7.16(a) *Curbs beam elements*

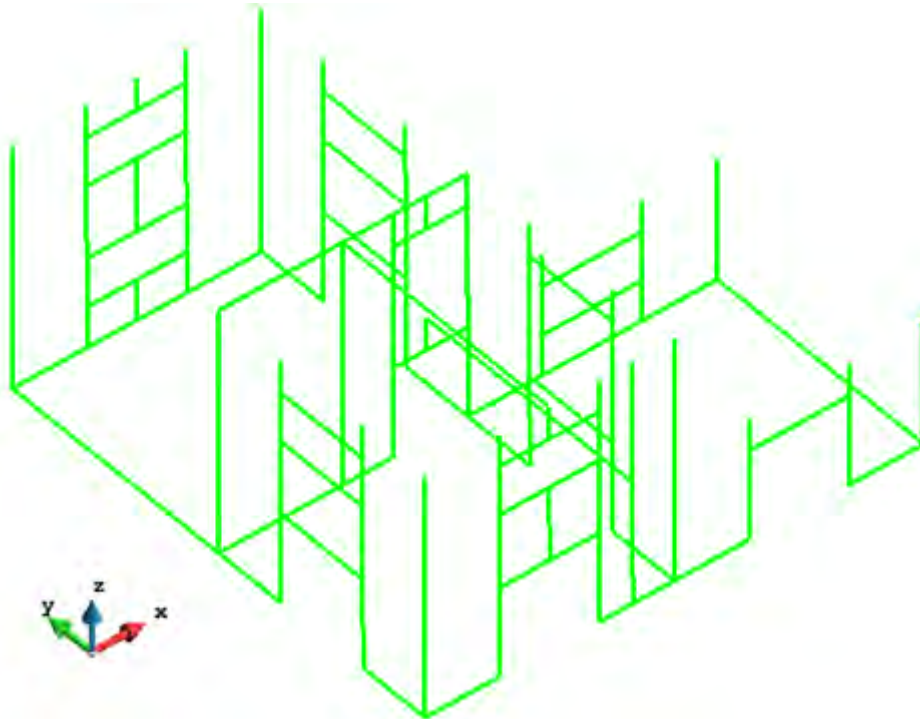


Figure 7.16(b) *Ordinary beam elements*

7.3.3 Connection properties

Many different connection properties are created to assign to each panel the previously calculated connections. Obviously, it is much more complicated to assign all the properties in the case of a real structure than in the Chapter 5 example, because there are a lot of lines and surfaces and it might be easy to forget a few. Moreover, since a property consists of hold-down and brackets, it is not so common that different lines have exactly the same property, therefore the connection properties are many. The “Custom-Stiffness” mode is exploited to have the possibility of changing the stiffness when desired, mostly to model the top lines of the vertical panels.

From an engineering point of view, the modelling is not so easy like just calculating the connections as in Section 7.2.4. In addition to the sides on which the hold-down and brackets are calculated with the preliminary design, it is important to think about the modelling of the other sides. Some simplifications are used in order not to overly complicate the problem, since it is just an example not aimed at a real project, therefore:

- The vertical lines (in z-direction) of the panels that represent the walls are modelled with infinite stiffness instead of calculating a distributed nailing: it is used the “Continuity connection” property. In general, a connection property with a distributed axial stiffness would be created, and it could be assigned to the vertical line of one panel; a “Continuity connection” property could be assigned to the same line of the other panel. This implicates to assign all the actual stiffness to one line and guarantee continuity to the other one, considering that the stiffness are then added, belonging to the same line. This is suitable because, being the springs in series, the force is distributed equally on the springs and the stiffness sum as $k_{\text{tot}} = 1/k_1 + 1/k_2$.
- The top lines of the vertical panels are modelled with a distributed axial stiffness; the hold-down axial stiffness is set to zero, while the parallel and orthogonal shear stiffness are infinite (a great value equal to 10^{12} N/m). Setting to zero the hold-down axial stiffness, but assigning a distributed axial stiffness, the latter distributes along the entire top line of the panel, including extreme

nodes. The same lines belonging to the slabs are modelled with all the stiffness equal to infinite (“Continuity connection” property). Thus, to simulate that the slab is not really infinitely rigid. An example of this modelling is represented by line 36 belonging to surfaces 6 and 31, shown in Figure 7.18. As belonging to surface 31, which represents the slab, line 36 has the “Continuity connection” property; conversely, as belonging to surface 6, which represents the wall, it has the property shown in Figure 7.17.

- In the case of surfaces which are actually a single panel, but they are represented by two different surfaces, the hold-down is only disposed on one side and not the other, corresponding to the side that flanks the other surface. An example of this modelling is represented by lines 4 and 5 belonging to surfaces 4 and 5, respectively, shown in Figure 7.18. The two surfaces are actually a unique wall, yet they are drawn with two different surfaces, to identify the centre of gravity of the first storey. This means that actually there is a hold-down on the left side of line 4 (on the point with less coordinate respect to the x-axis) and one on the right of line 5 (on the point with greater coordinate respect to the x-axis). Figure 7.19 shows the connection properties of the two lines: line 4 has only hold-down 1 (with less coordinate), while line 5 has only hold-down 2 (with greater coordinate). The brackets are calculated as usually, depending on the length of the line; the distributed axial stiffness and the orthogonal shear stiffness are set to a small value equal to 100 N/m.

```

Mode: Stiffness
Stiffness
  Axial stiffness HD1: 0
  Axial stiffness HD2: 0
  Distributed axial stiffness: 6.71e+7
  Distributed parallel shear stiffness: 1.0e+12
  Distributed orthogonal shear stiffness: 1.0e+12
  Rk HD1: 0
  Rk HD2: 0
  Rk distributed: 2.68e+4
  Correction factor kmod: 1.1
  Material security coefficient: 1.3

```

Figure 7.17 Connection property of line 36 belonging to surface 6

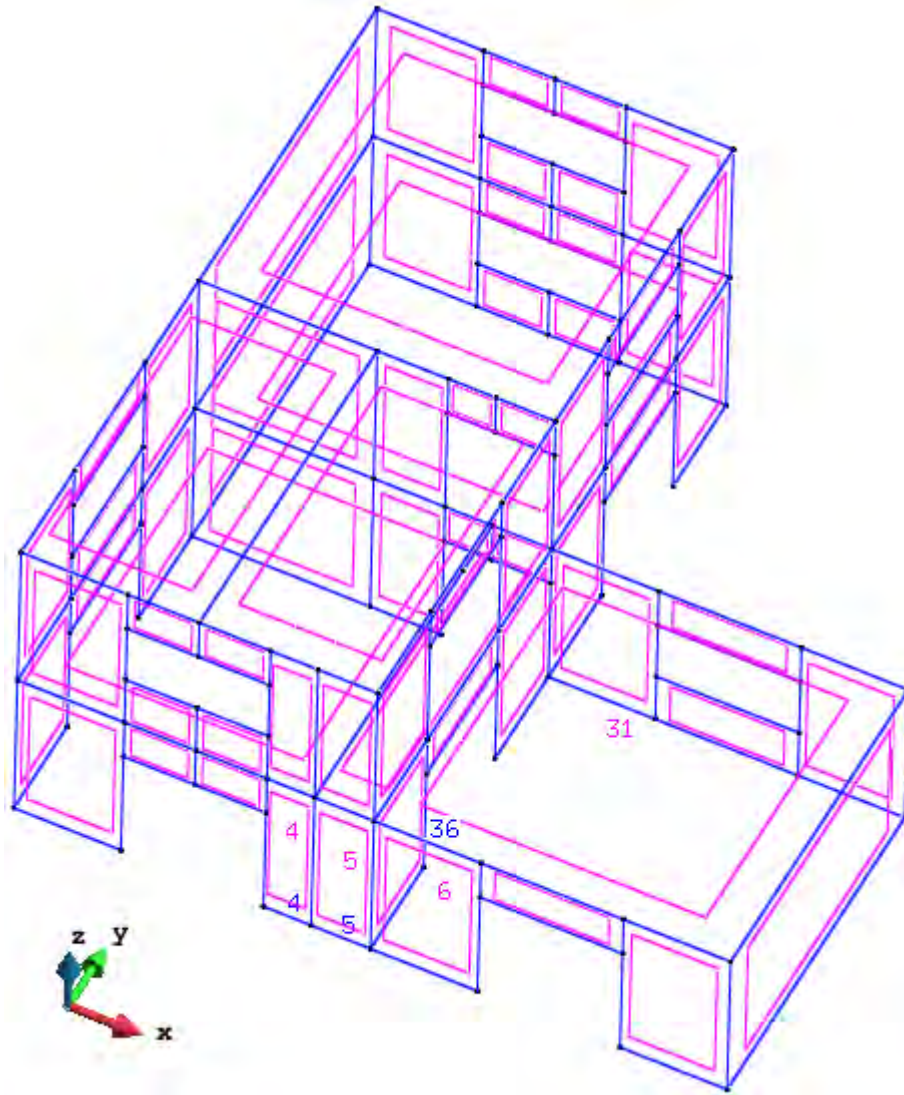


Figure 7.18 Example of line (36) belonging to wall (surface 6) and slab (surface 31); Example of lines (4 and 5) belonging to walls (surfaces 4 and 5) which are actually a single wall

<ul style="list-style-type: none"> 4_4 Mode: Stiffness Stiffness <ul style="list-style-type: none"> Axial stiffness HD1: 3.49e+8 Axial stiffness HD2: 0 Distributed axial stiffness: 100 Distributed parallel shear stiffness: 6.04e+7 Distributed orthogonal shear stiffness: 100 Rk HD1: 2.01e+5 Rk HD2: 0 Rk distributed: 2.68e+4 Correction factor kmod: 1.1 Material security coefficient: 1.3 	<ul style="list-style-type: none"> 5_5 Mode: Stiffness Stiffness <ul style="list-style-type: none"> Axial stiffness HD1: 0 Axial stiffness HD2: 3.49e+8 Distributed axial stiffness: 100 Distributed parallel shear stiffness: 6.04e+7 Distributed orthogonal shear stiffness: 100 Rk HD1: 0 Rk HD2: 2.01e+5 Rk distributed: 2.68e+4 Correction factor kmod: 1.1 Material security coefficient: 1.3
---	---

Figure 7.19 Connection properties of lines 4 and 5 belonging respectively to surfaces 4 and 5

7.3.4 Boundary conditions and loads

For the analysis of the structure all the displacements and rotations of the bottom lines are restrained.

Relatively to the loads, considering that the seism can act in any direction, two different analyses are performed: one with the seismic forces applied in x-direction and the 30% of the same forces applied in y-direction, another the other way around (see Section 7.2.3).

Acting the seism in x-direction (configuration A), the forces that should be applied at the barycentre of the storeys are:

$$W_{R_X} = 225 \text{ kN} \quad ; \quad W_{R_Y} = 67.5 \text{ kN}$$

$$W_{F_X} = 239 \text{ kN} \quad ; \quad W_{F_Y} = 71.7 \text{ kN}$$

Acting the seism in y-direction (configuration B), the forces that should be applied at the barycentre of the storeys are:

$$W_{R_X} = 67.5 \text{ kN} \quad ; \quad W_{R_Y} = 225 \text{ kN}$$

$$W_{F_X} = 71.7 \text{ kN} \quad ; \quad W_{F_Y} = 239 \text{ kN}$$

In addition to these forces, in both analyses the seismic loads of the roof and floor and the walls loads are also applied (see Section 7.2.1). All the forces and loads are applied as uniformly distributed, to avoid concentrations of stresses.

7.4 Results

Meshing the geometry and launching the calculation, the structure can be analysed. The results are shown in the following sections, starting from the discretization, up to the displacement, tension and reactions fields in the two load configurations analysed.

7.4.1 Structure discretization

A structured mesh is used for the discretization of the structure. The dimension of each meshed element is equal to 400 mm, for a total amount of 2976 quadrilateral elements, 910 linear elements and 3059 nodes (Figure 7.20).

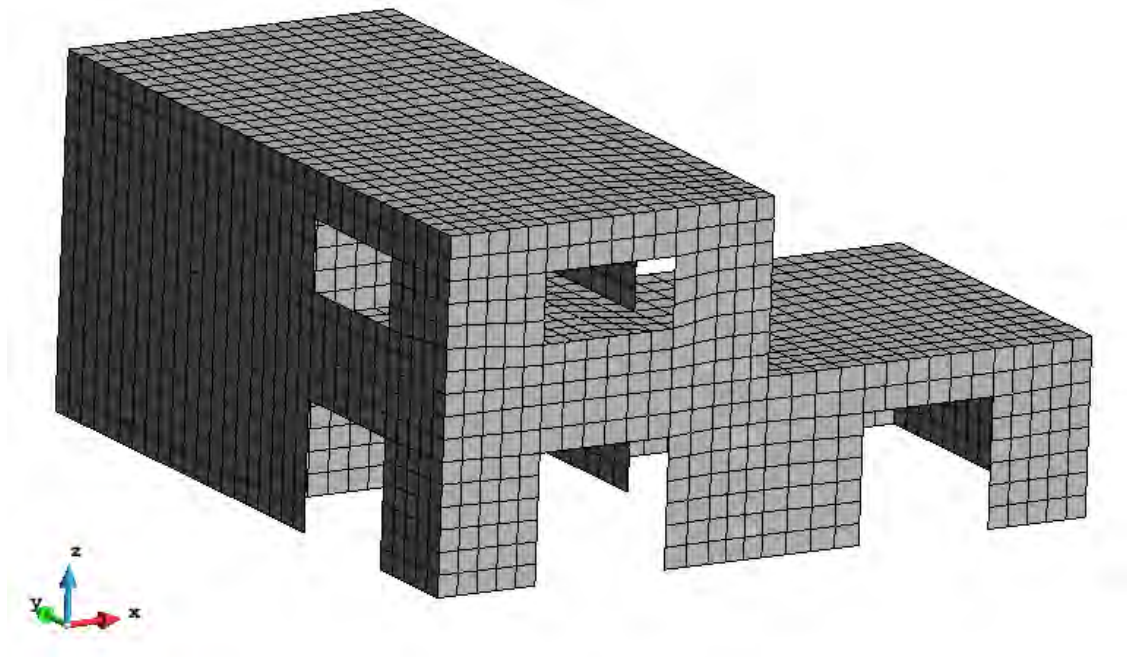


Figure 7.20 *Meshed view of the structure in GiD*

After the application *Xlam-driver* comes into play, the discretization used leads to a different number of elements and nodes. The system increases the number of nodes to 5103 and the number of elements to 6215, 2329 of which are spring elements.

7.4.2 Displacement field

The displacement field for the two load configurations and the total displacement of the structure, taking only into account the storeys seismic forces, are shown in Figures 7.21/7.25.

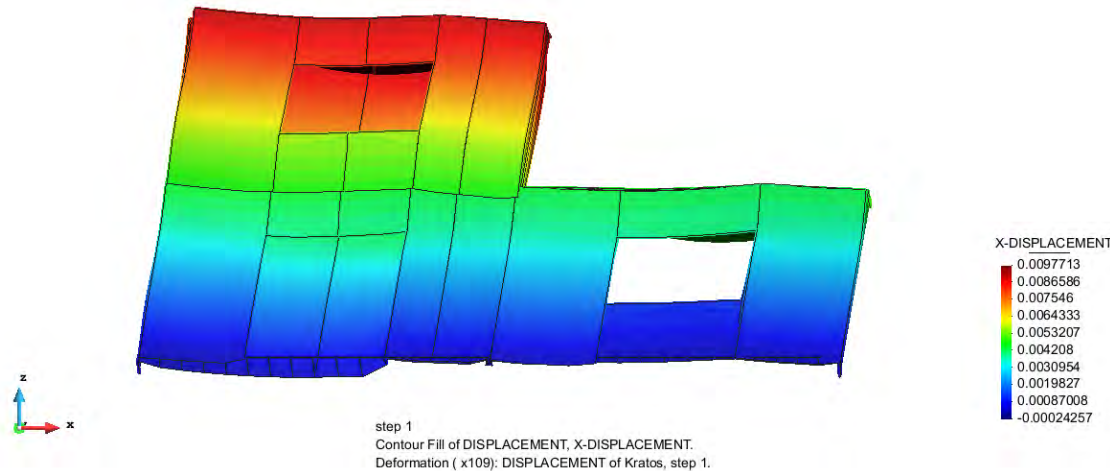


Figure 7.21 X-displacement contour [m] for the configuration A

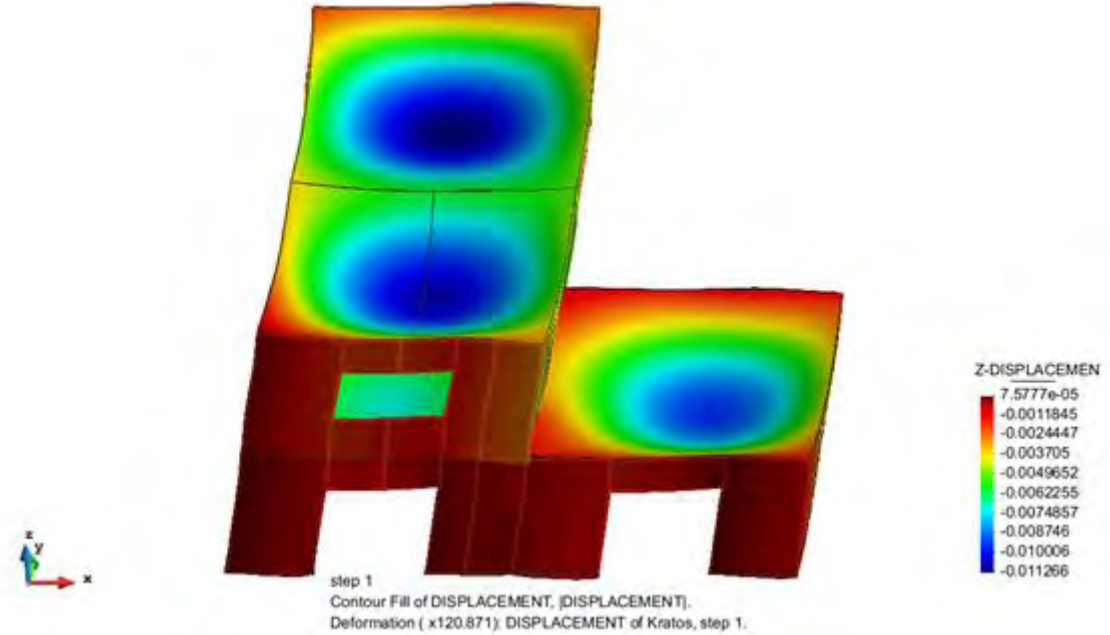


Figure 7.22 Z-displacement contour [m] for the configuration A

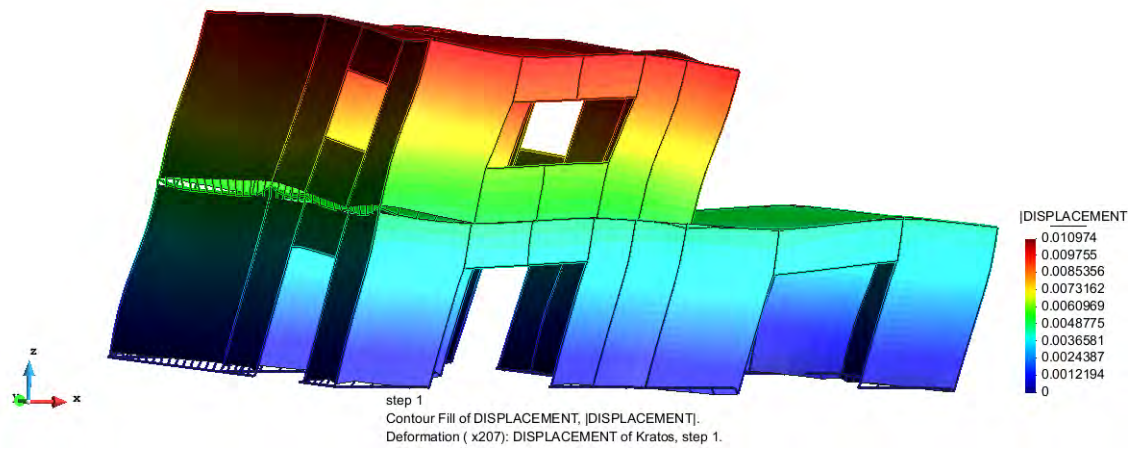


Figure 7.23 Contour of the absolute value of the displacement[m] for the configuration A, taking into account only the storeys seismic forces

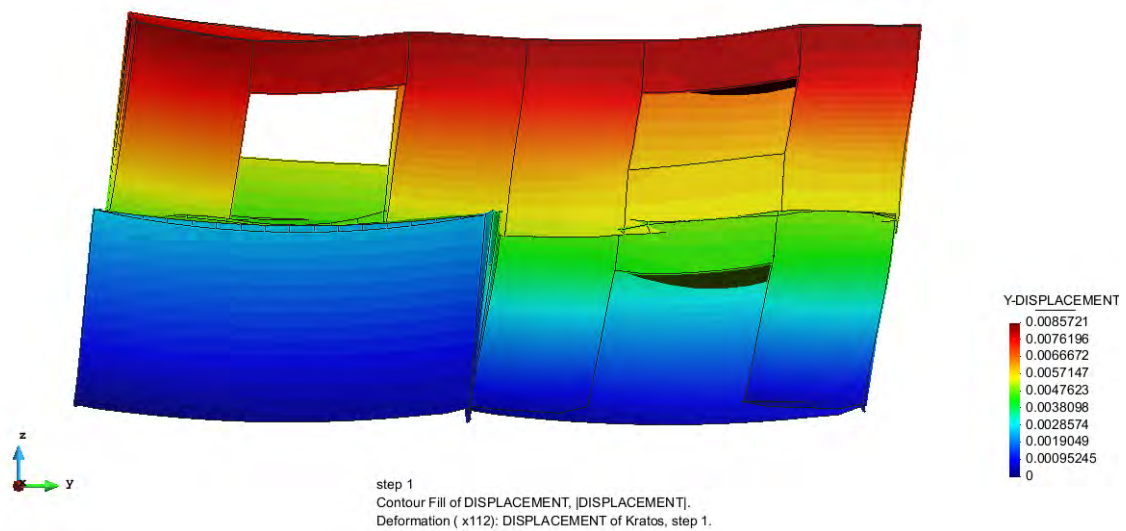


Figure 7.24 Y-displacement contour [m] for the configuration B

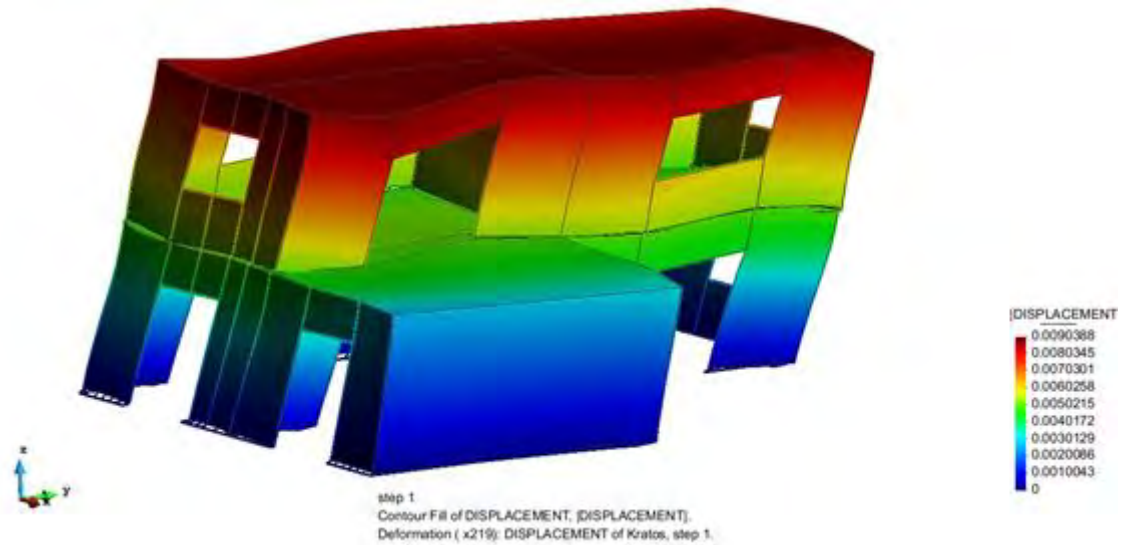


Figure 7.25 Contour of the absolute value of the displacement[m] for the configuration B, taking into account only the storeys seismic forces

Figures 7.21, 7.22 and 7.24 show the displacement of the structure subjected to all the load conditions. The building results quite rigid, since the maximum displacements are in the order of magnitude of 1 cm. In Figures 7.23 and 7.25 the structure is only subjected to the seismic forces condition. The spring elements are clearly visible; the hold-down springs behave correctly, since each wall tends to rigidly rotate around its mid-point of the lower side.

7.4.3 Tension field

The tension field for the two load configurations and taking only into account the storeys seismic forces are shown in Figures 7.26/7.31. The shell forces are actually displayed, rather than the tensions, meaning that they are already integrated through the shell thickness.

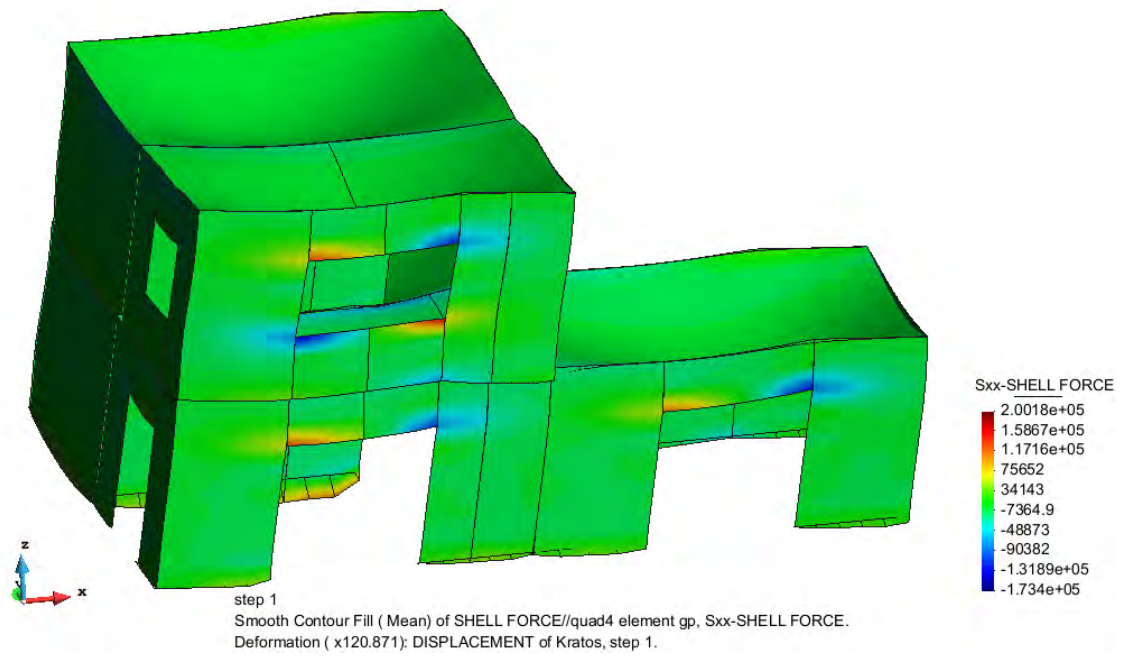


Figure 7.26 Shell Force S_{xx} contour [N/m] for the configuration A

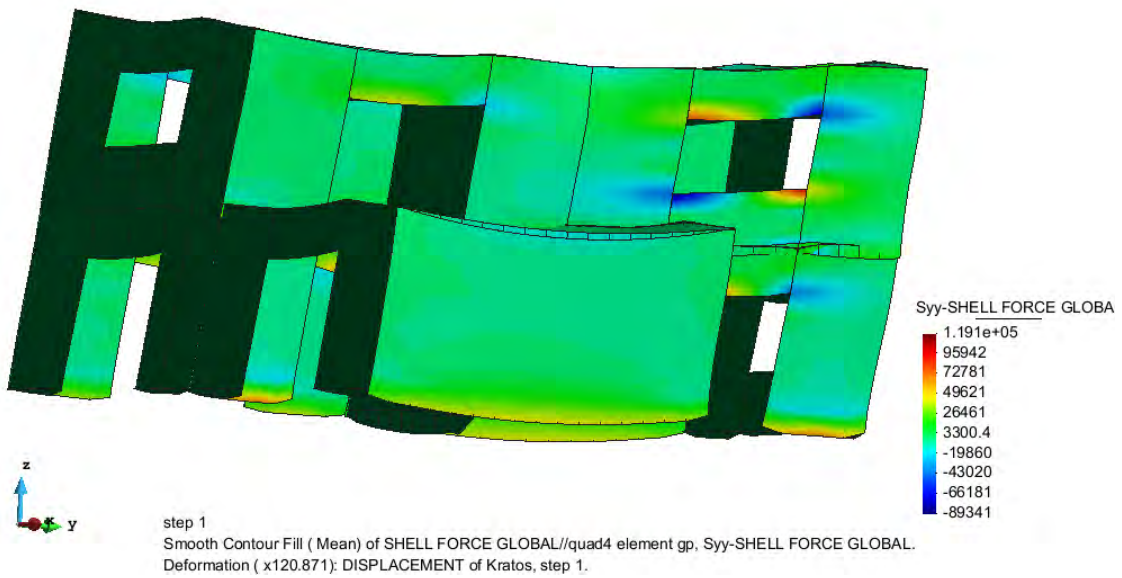


Figure 7.27 Shell Force S_{yy} contour [N/m] for the configuration A

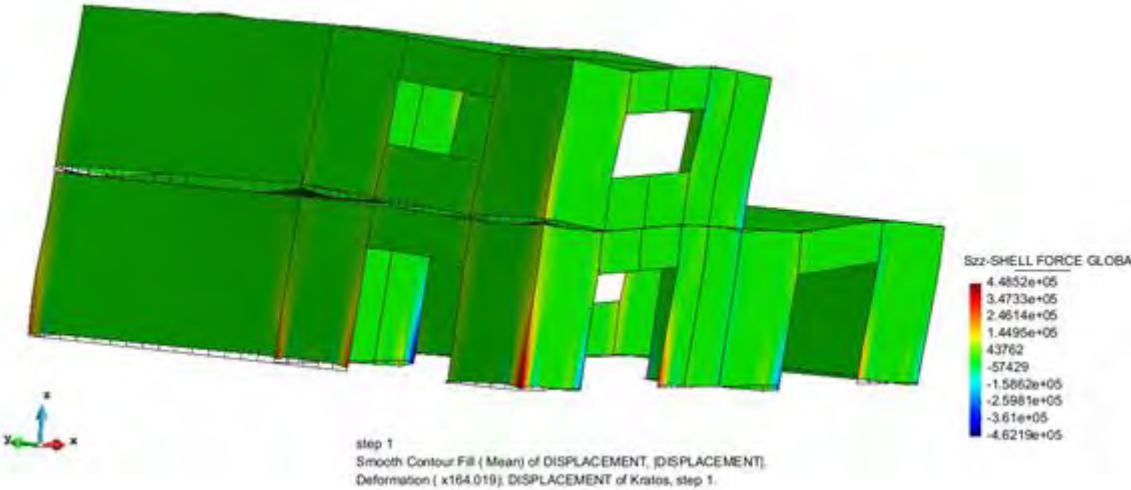


Figure 7.28 Shell Force S_{zz} contour [N/m] for the configuration A, taking into account only the storeys seismic forces

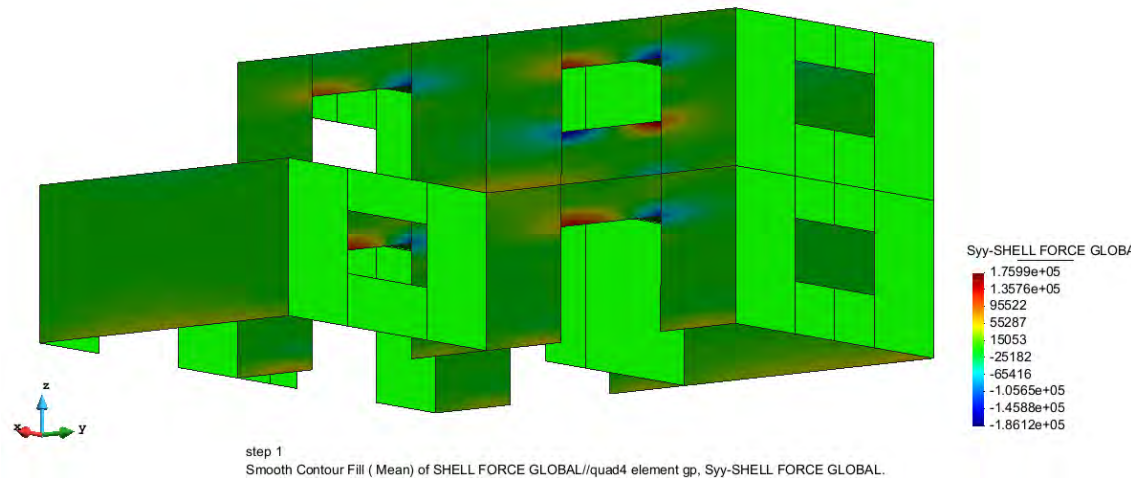


Figure 7.29 Shell Force S_{yy} contour [N/m] for the configuration B

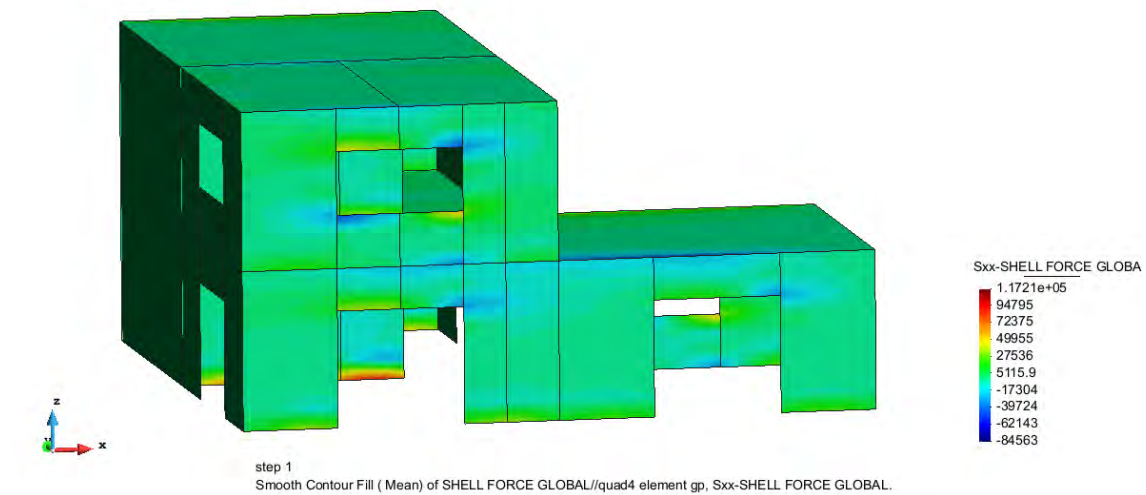


Figure 7.30 Shell Force S_{xx} contour [N/m] for the configuration B

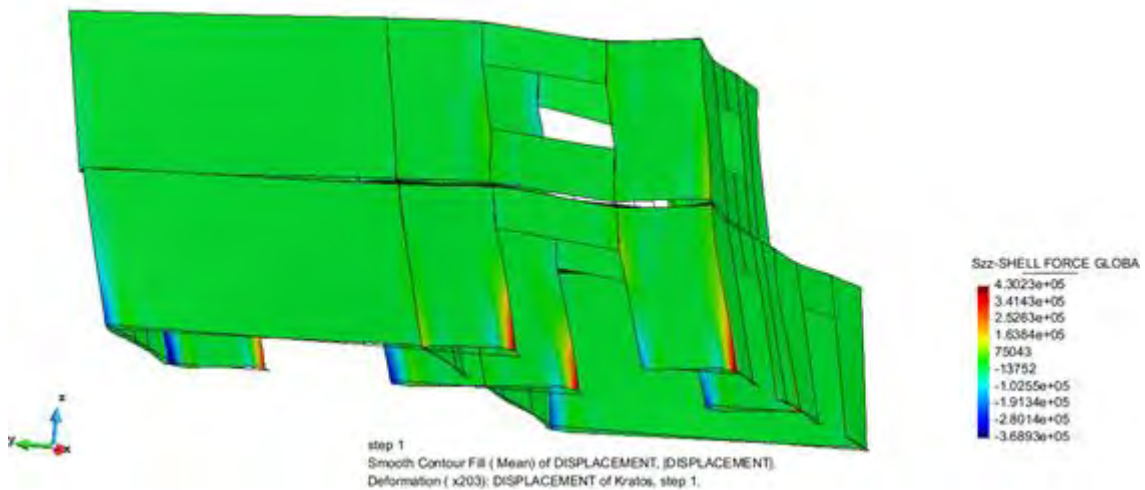


Figure 7.31 Shell Force S_{zz} contour [N/m] for the configuration B, taking into account only the storeys seismic forces

Figures 7.26, 7.27, 7.29 and 7.30 show the shell forces of the structure subjected to all the load conditions. The peaks of tension are discoverable at the windows corners. In Figures 7.28 and 7.31 the structure is only subjected to the seismic forces condition. The peaks of tension are correctly localized at the hold-down springs; the compression and tension zones follow correctly the deformation of the structure.

7.4.4 Reactions

The reactions vector field is shown in Figures 7.32/7.37; it could be a check of the correct behaviour of the structure. The Z-reactions vector field is shown for the two load configurations and taking only into account the storeys seismic forces, to visualise the hold-down reactions. The X-reactions vector field and the Y-reactions vector field are also shown, respectively for the configurations A and B.

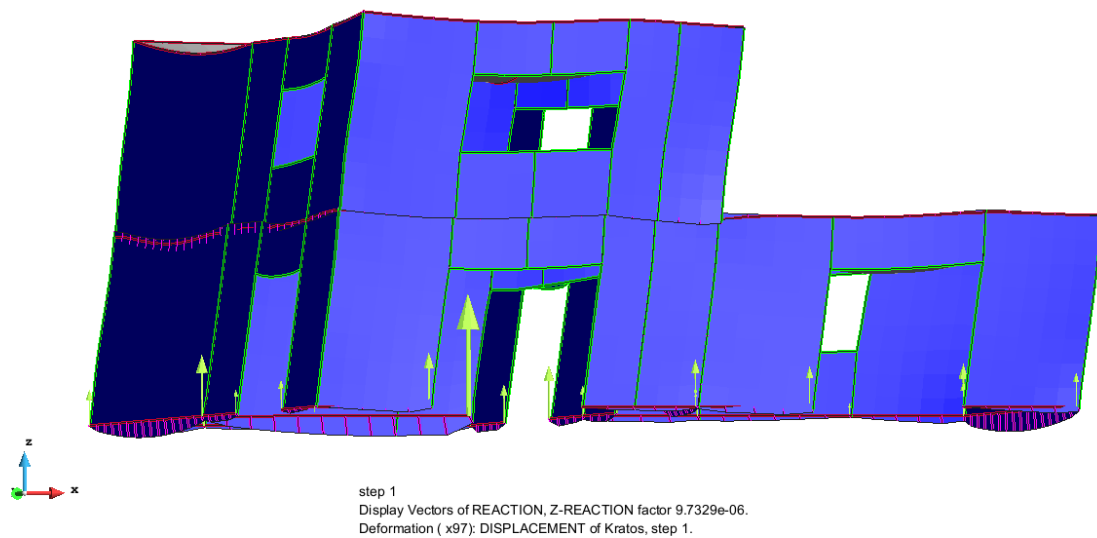


Figure 7.32 Z-Reaction vectors display for the configuration A

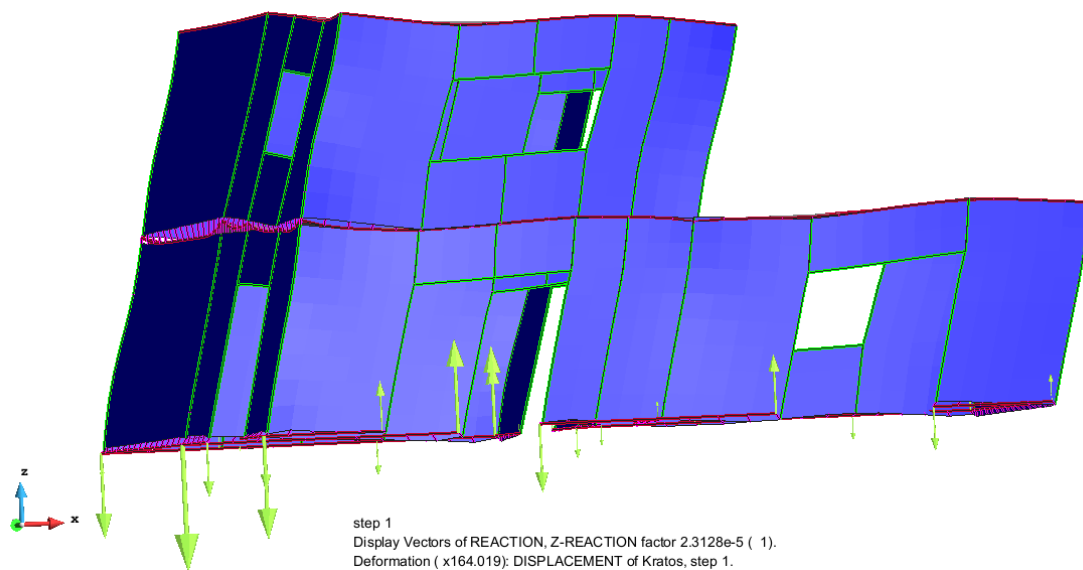


Figure 7.33 Z-Reaction vectors display for the configuration A, taking into account only the storeys seismic forces

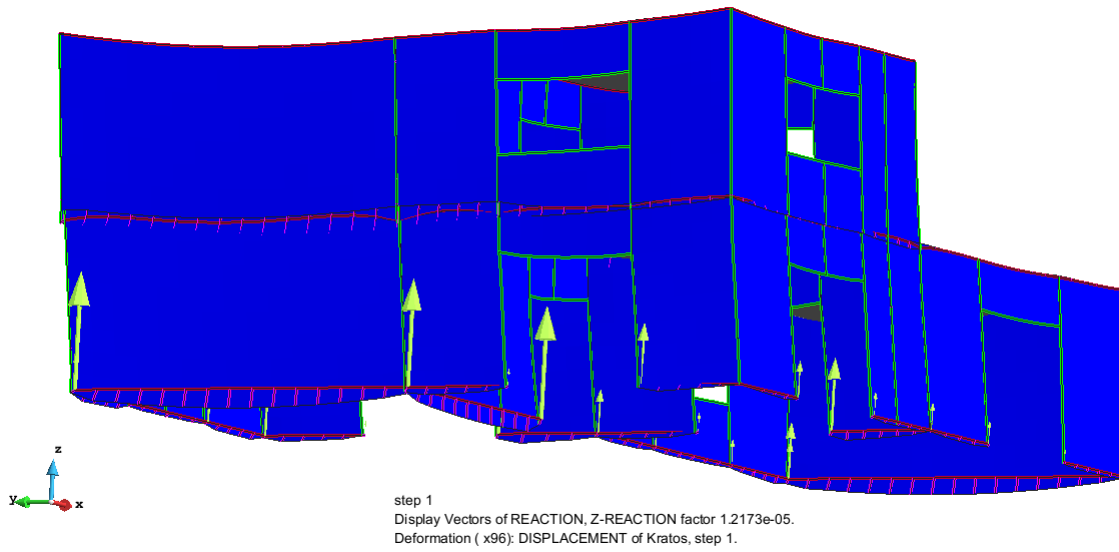


Figure 7.34 *Z-Reaction vectors display for the configuration B*

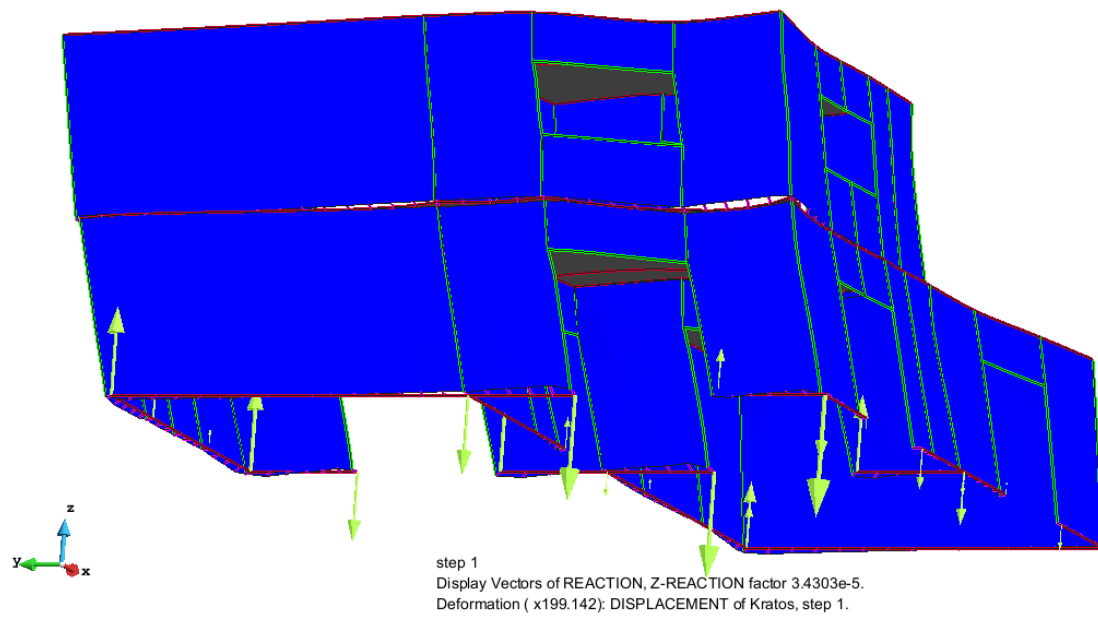


Figure 7.35 *Z-Reaction vectors display for the configuration B, taking into account only the storeys seismic forces*

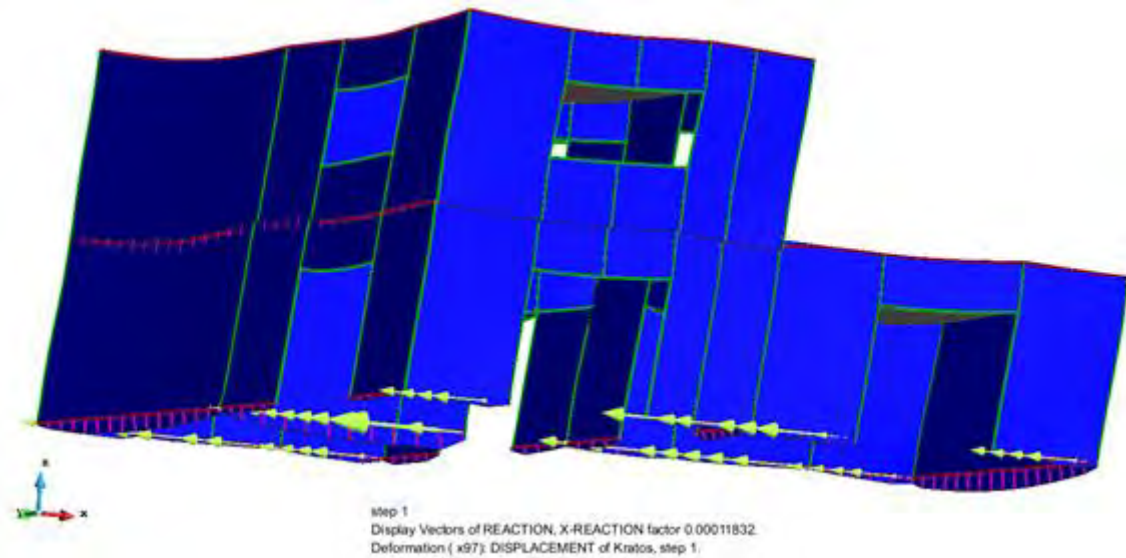


Figure 7.36 X-Reaction vectors display for the configuration A

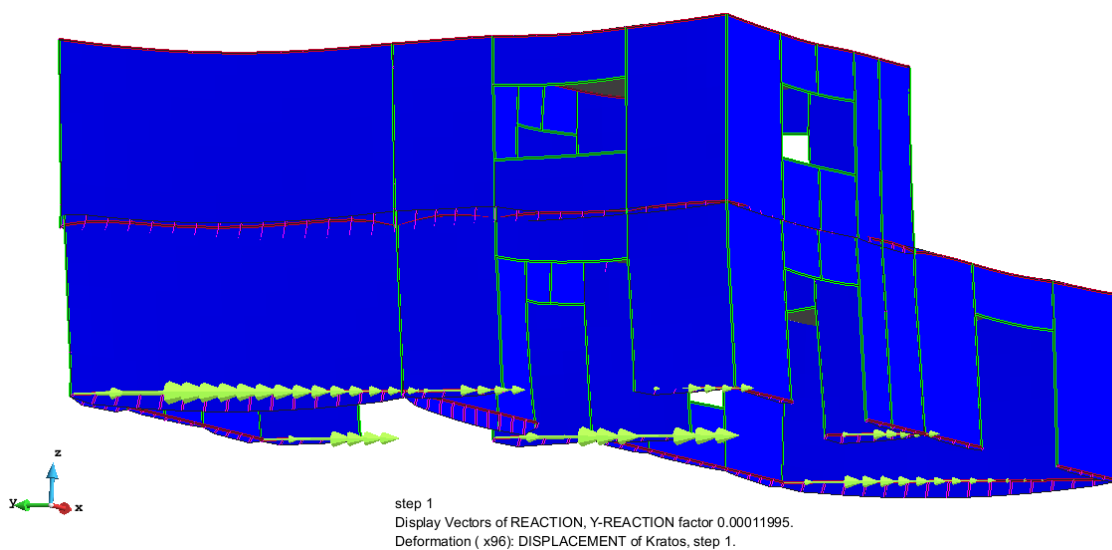


Figure 7.37 Y-Reaction vectors display for the configuration B

The reactions vector field is a reply of the correct functioning of the software. Figures 7.32/7.35 testify that the hold-down axial stiffness and the distributed axial stiffness are assigned to the correct springs because only the ones set as hold-down present a vertical reaction. Figures 7.36 and 7.37 testify that the distributed parallel and orthogonal shear stiffness are correctly assigned because only the springs allocated on the sides in direction parallel to the seismic action presents an horizontal reaction.

CHAPTER 8

CONCLUSION

8.1 Main contributions

Within the scope of the research project, the pre-process and analysis phases of a software for the analysis, calculation, design and verification of X-Lam structures have been performed. The pre-process is discussed in this thesis, while the analysis phase is detailed in “*An algorithm for numerical modelling of Cross-Laminated Timber structures*” by Gabriele D’Aronco. The key objective of this research work was to allow the automatic modelling of connections between X-Lam panels, lying in the connections the modelling of CLT buildings. To achieve the goal, a new problem type for GiD interface and a new application for KRATOS framework have been performed.

The primary conclusions about the developed phases focus on the achieved goals within the scope of the whole research project. The main contribution is the development of a strategy for numerical modelling of cross-laminated timber structures. Found the strategy, sound bases for the pre-process and analysis phases of the software have been laid.

The pre-processor allows to assign the connection properties, the material and the elements properties, as well as the classical conditions of all other problem types. The panels are modelled as orthotropic, with a cross section composed by layers which can have different thickness (and, obviously, different grain orientation). The post-process phase is already pre-set, relatively to the iterative calculation of optimal connections and the verification of connections and panels. The examples shown in the thesis “*An algorithm for numerical modelling of Cross-Laminated Timber structures*” by Gabriele D’Aronco, testify that the algorithm implemented for the automatic modelling of the connections works as expected and the connections stiffness are actually correctly assigned to the springs. Therefore the behaviour of the structure can

be considered reliable, within the limits imposed by linear elastic analysis. The analysis works as expected for each kind of CLT structure.

The goals of these phases of the research project have been achieved partly, developing a strategy that allows the automatic modelling of connections between X-Lam panels. However, the developed phases of the software are still not perfectly ready, being available improvements aimed at a better end result. The large probability of error of the other commercial software due to the need of manually duplicate the nodes and create the spring elements one by one in the interface is drastically reduced by this new software, which does it automatically. The modelling of an X-Lam structure is now easier, faster and safer thanks to the reduced possibility of human errors.

8.2 Recommendations for further research

While this thesis, together with “*An algorithm for numerical modelling of Cross-Laminated Timber structures*”, tries to lay sound bases for the pre-process and analysis phases of the software, a significant amount of further research is required for the development of the post-process and verification phases, in addition to some improving about the already developed phases. Particularly, some specific aspects that could be improved about the pre-process and analysis phases are:

- Setting or calculation (depending on the mode) of the stiffness of the distributed springs in N/m^2 instead of N/m , meaning setting the distance between the brackets or the nails instead of their number. This improvement needs the splitting of the hold-down connection property from the brackets or distributed nailing one, because currently the combination of the two connections is assigned to each line.
- Leaving the possibility to choose a different material for each layer of the panel. This improvement will be possible with the new version of Kratos, which will present a refined structure of the interface.
- Adaptation of the orthotropic shell to inclined surfaces, meaning surfaces not lying on X-Y, X-Z or Y-Z axes.

- Removal in the *x-lam driver* application of the restraint about the z-axis as vertical one: thus, the hold-down springs will also be created on the vertical beams.
- Implementation of other types of linear analysis (response history analysis, response spectrum analysis) in the Kratos framework, to allow the calculation of non-regular and more complex structures.
- Implementation of non-linear constitutive law for the spring elements that model the connections, to enable the simulation of the contact problem. Thus, the use of non-linear static (pushover) and non-linear time history (dynamic) analyses will also be available.

Relatively to the post-process and verification phases, recommendations for further research are outlined, to develop the missing phases of the research project:

- Development of a procedure for the automatic verification of connections and panels according to the European codes. This phase is already pre-set by the pre-process because the design load-carrying capacities of the connections are already calculated and printed in the file “*More-Connections*”, as well as the strength values of the material of the panels are printed in the file “*More-Materials*”.
- Development of a procedure for the iterative calculation of the optimal connections, once set first attempt solution. The iterative nature of the design is not only a peculiarity of the X-Lam structures, but it is a particular feature of the design in general. This peculiarity is immediately realised recalling that the intensity of the seismic action depends, through the design spectrum, on the periods of the fundamental modes of vibration of the structure, which in turn depend significantly on the stiffness of the connections. At each iteration, the optimal connections necessary to resist to the active external forces can be calculated. Nonetheless, external forces are referred to the stiffness distribution of the previous iteration, therefore the connections can be updated in the model up to their convergence with the dichotomous method (see “*Una procedura numerica per il progetto di edifici in X-Lam*” by Massimiliano Zecchetto).

REFERENCES

- Augustin, M. (2008). *Timber structures - Handbook 1 of Educational materials for designing and testing of timber structures: TEMTIS*. Leonardo da Vinci Pilot Project No. CZ/06/B/F/PP/168007. Ostrava, Czech Republic: VSB-Technical University of Ostrava.
- Bathon L.A., Bletz O. (2006). *Long term performance of continuous wood-concrete-composite systems*. 9th World Conference on Timber Engineering, Portland, Oregon, USA.
- Briani A., Simeone P., Ceccotti A. (2012). *MAI, IVALSA modular house*. Proceedings of the 12th World Conference on Timber Engineering, Auckland, New Zealand.
- D'Aronco G. (2015). *An algorithm for numerical modelling of Cross-Laminated Timber structures*. Padova, Italy
- D.M. 14-01-2008. New Technical Standards for Construction. CSLP, Roma, Italy.
- EN 338 (2004). Strength classes for solid wood. CEN, Brussels, Belgium.
- EN 1194 (1999). Timber structures - Glued laminated timber - Strength classes and determination of characteristic values. CEN, Brussels, Belgium.
- EN 12524 (2000). Building materials and products – Hygrothermal properties – Tabulated design values. CEN, Brussels, Belgium.
- EN 1993-1-1 (2005). Eurocode 3: Design of steel structures - Part 1-1: General rules and rules for buildings. CEN, Brussels, Belgium.
- EN 1995-1-1 (2009). Eurocode 5: Design of timber structures – Part 1-1: General rules and rules for buildings. CEN, Brussels, Belgium.
- EN 1998-1 (2004). Eurocode 8: Design of structures for earthquake resistance – Part 1: General rules, seismic actions and rules for buildings. CEN, Brussels, Belgium.

ETA-06/0138 (2006). European technical approval: KLH solid wood slabs. European Organisation for Technical Approvals, Brussels, Belgium.

FPIinnovations (2013). *CLT Handbook: Cross-laminated timber*, US Edition.

Frangi A., Bochicchio G., Ceccotti A., Lauriola, M.P. (2008). *Natural Full-Scale Fire Test on a 3 Storey XLam Timber Building*. 10th World Conference on Timber Engineering, Miyazaki, Japan.

Gavric I. (2012). *Seismic behaviour of Cross-Laminated Timber Buildings*. Trieste, Italy.

Gavric I., Fragiaco M. e Ceccotti A. (2014) *Cyclic behaviour of typical metal connectors for cross-laminated (CLT) structures*.

Gavric I. e Popovski M. (2014). *Design models for CLT shearwalls and assemblies based on connection properties*.

GiD. <http://www.gidhome.com>

Giordano G. (1993). *Tecnica delle costruzioni in legno*, Hoepli, Milano.

KLH. <http://www.klh.at>

Kratos. <http://kratos-wiki.cimne.upc.edu/index.php>

Kratos. <http://www.cimne.com/kratos>

Martinson. <http://www.martinsongroup.com>

Piazza M., Tomasi R. e Modena R. (2007). *Strutture in legno*, Hoepli, Milano.

Popovski M., Schneider J., Schweinsteiger M. (2010). *Lateral load resistance of cross-laminated wood panels*. Proceedings of the 11th World conference on timber engineering, Riva del Garda, Italy.

Python. [https://en.wikipedia.org/wiki/Python_\(programming_language\)](https://en.wikipedia.org/wiki/Python_(programming_language))

Rothoblaas. <http://www.rothoblaas.com>

Sihga. <http://www.sihga.com/en/products/idefix/idefix-ifs.html>

Stora Enso. <http://www.storaenso.com>

Tcl. <http://en.wikipedia.org/wiki/Tcl>

Tcl. <http://zetcode.com/lang/tcl/tcl>

Tcl. https://en.wikibooks.org/wiki/Tcl_Programming/Introduction

Timber Tower. <http://www.timbertower.de>

Traetta G. (2007). *Connection Techniques for CLT elements*. Temtis Seminar: Cross-Laminated Timber, Graz, Austria.

Wood Works. <http://www.wood-works.ca>

Yates M., Linegar M., Dujic B. (2008). *Design of an 8 storey Residential Tower from KLH – Cross Laminated Solid Timber Panels*. Proceedings of the 10th World Conference on Timber Engineering, Miyazaki, Japan.

Zecchetto M. (2015). *Una procedura numerica per il progetto di edifici in X-Lam*. Padova, Italy.